SMS User Manual (v11.1)

Surface-water Modeling System

PDF generated using the open source mwlib toolkit. See http://code.pediapress.com/ for more information.
PDF generated at: Wed, 06 Feb 2013 23:29:54 UTC

Contents

Articles

References

1. Introduction

What is SMS?

SMS (Surface-water Modeling System) is a complete program for building and simulating surface water models. It is a graphical user interface and analysis tool that allows engineers and scientists to visualize, manipulate, analyze, and understand numerical data and associated measurements. Many of the tools in SMS are generic. They are designed to facilitate the establishment and operation of numerical models of rivers, coasts, inlets, bays, estuaries, and lakes. It features 1D and 2D modeling and a unique conceptual model approach. Some of the currently supported models in SMS include ADCIRC, BOUSS-2D, CGWAVE, CMS-Flow, CMS-WAVE (WABED), FESWMS, GenCade, PTM, STWAVE, TABS, and TUFLOW.

Introduction to SMS

- The Highlights provide a summary of SMS capabilities.
- The SMS Tutorials are step-by-step guides for building models and using SMS features. They are an excellent place to begin learning how to use SMS.
- See Layout of the Graphical Interface for more information on the organization of the toolbars, menus, and windows in SMS.
- Much of the SMS functionality is divided into Modules based upon the type of data (grids, meshes, GIS, etc). SMS also contains features that are not tied to specific modules.
- SMS supports a number of Numerical Models with a variety of uses including hydraulics, wave modeling, and particle tracking.

History

SMS was initially developed by the Engineering Computer Graphics Laboratory (later renamed in September, 1998 to Environmental Modeling Research Laboratory or EMRL) at Brigham Young University [1] in the late 1980s on Unix workstations. The development of SMS was funded primarily by The United States Army Corps of Engineers [2]. It was later ported to Microsoft Windows platforms in the mid 1990s and support for HP-UX, IRIX, DEC-OSF, and Solaris platforms was discontinued.

In April 2007, the main software development team at EMRL entered private enterprise as Aquaveo LLC [3]. and continue to develop SMS and other software products, such as WMS (Watershed Modeling System) and GMS (Groundwater Modeling System).

References

[1] http://home.byu.edu/home/
[2] http://www.usace.army.mil/
[3] http://www.aquaveo.com/

Tutorials

A rich set of step-by-step tutorials has been developed to aid in learning how to use SMS.

Tutorial Installation

There are two options for installing tutorials. You can download the tutorials and files individually by subject matter in the SMS learning center or you can download an install that includes all of the core tutorials and files. Some of the additional tutorials (TUFLOW, PTM, CGWAVE) still need to be downloaded separately. Since most users only need a subset of the tutorials, the recommended approach is to download and install them as needed from the SMS learning center.

Accessing Tutorials Through the SMS Learning Center

The SMS Learning Center is the portal to all of the training you will need to learn how to use SMS. It is there that you can gain access to all of the SMS tutorial documents and files via download. To access the SMS Learning Center and tutorials click here [1], or follow these steps:

- Go to www.aquaveo [3]
- Click on the SMS logo found on the right side of the screen. This will take you to the SMS homepage.
- Once at the SMS homepage, click on the SMS Learning Center Icon.
- In the SMS Learning Center page, scroll down to the tutorials. There, the tutorials are divided into two groups: general SMS tutorials, and tutorials for specific models found within SMS. All are available for download.

Opening and Downloading Tutorials

Each tutorial consists of a PDF document, and its associated tutorial files.

- Open the PDF document by right clicking on it, then selecting Open link in new tab.
- Then, click on the files icon and choose the Save as.. option. *You can choose the directory in which you would like to run and save your tutorial files.
- Once the files have finished downloading, select the Open option.
- When the data files come up, you must extract the files by clicking on the Extract all files option.
- Once the files are extracted, you are ready to begin with the tutorial.

°*Note: Do not save the tutorial documents and files to the Program Files directory.*

Related Topics

- Sample Problems

References

[1] http://www.aquaveo/sms-learning/

Sample Problems

In addition to the tutorial files, numerous test cases are available for download from the Aquaveo Verification Repository [1].

The Aquaveo Verification Repository is designed to store case studies which can be used to verify the accuracy and capabilities of various numeric models. The case studies contain within the repository will eventually help to build a selection tool. The selection tool will use numeric model results to suggest appropriate models to use for a study. The results will be determined by the performance of the model when faced with certain site characteristics.

The cases can be searched using the "Search" links found in the navigation menu. Each model type is contained in a separate repository. The search page will allow you to search for case studies containing particular attributes. Performing a search with no selections will allow you to browse all studies contained in the repository.

SMS users may add test cases to the repository. In order to add studies to the repository you must create an account. The provided contact information will not be released to anyone. We invite everyone to contribute to the repository. More cases means the models can be tested more thoroughly, which will result in a better selection tool.

Related Topics

- SMS Tutorial Files

External Links

- Aquaveo Verification Repository [1]
- Hollingsworth, Jason M (2008). Foundational Data Repository for Numeric Engine Validation. Thesis, Brigham Young University. [2]

References

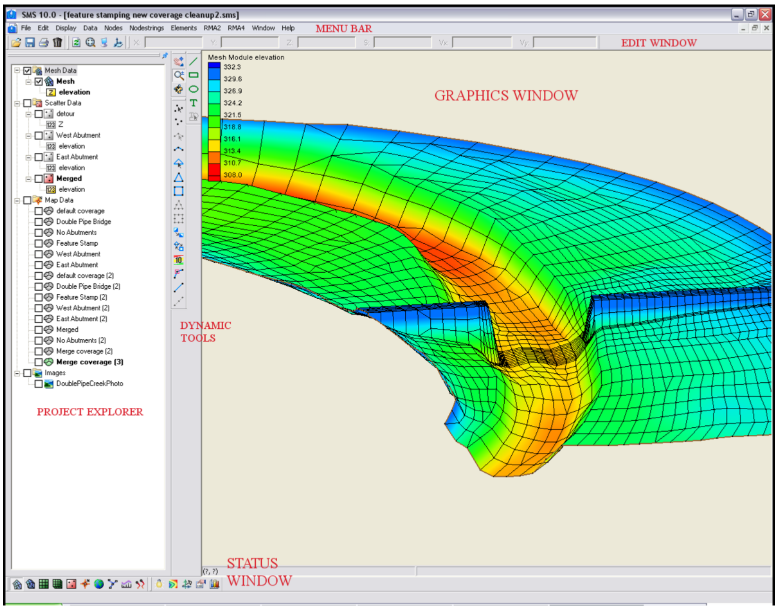
[1] http://verification.aquaveo.com/
[2] http://contentdm.lib.byu.edu/ETD/image/etd2661.pdf

Highlights

Layout

- The project explorer shows data currently loaded in project
- Menu bar depends upon the active module and model
- Edit window show x, y, z, scalar, and vector values
- Edit window values can be edited in some circumstances
- The status window on the bottom of the graphics window shows coordinates and selection information
- Help information is displayed at the bottom of the SMS screen
- Several toolbars are used in SMS. The dynamic tools change based upon the current module.

More Info...



SMS Layout

Modules

- Data is divided into modules based upon the data type


Module Toolbar

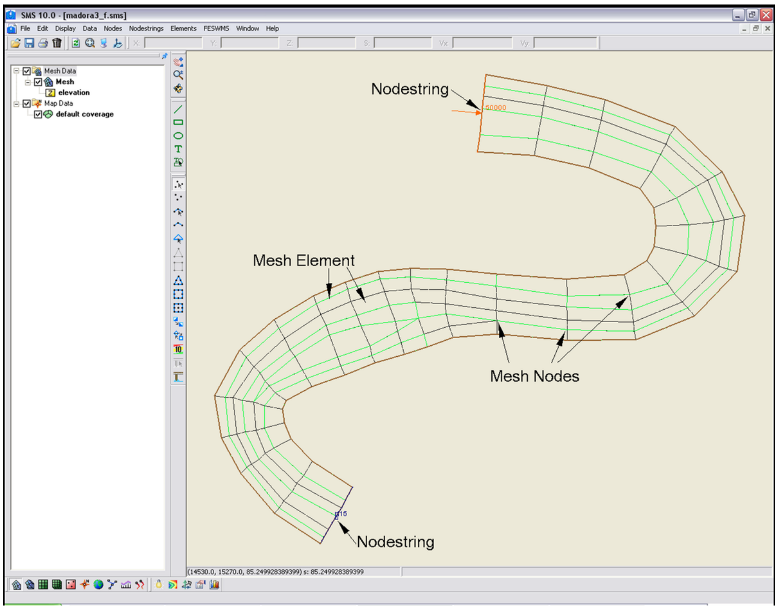
- There is always one active module
- The menus and toolbars are based upon the active module
- The current module may be selected in module bar or by selecting an object in project explorer

More Info...

Mesh Module

- Used to create, edit, and visualize mesh data
- Also referred to as unstructured grids or finite element meshes
- Meshes defined by nodes and elements
- Several element types are supported

More Info...

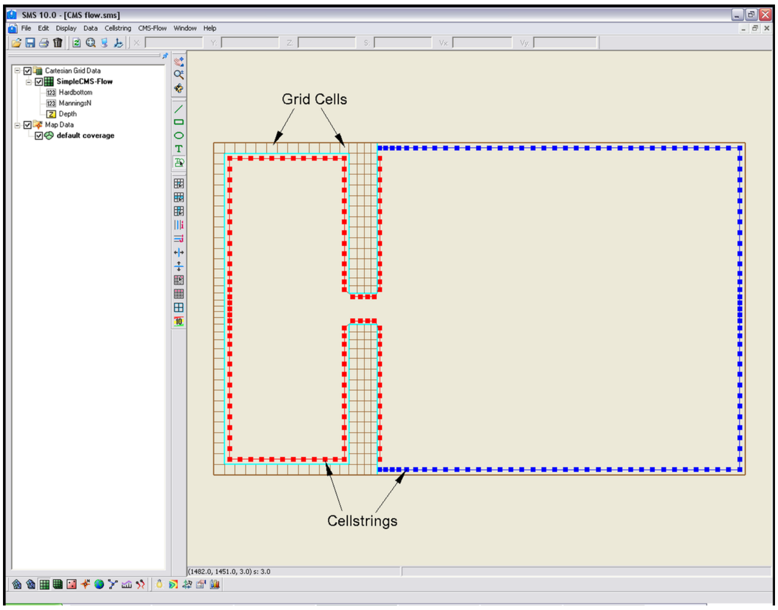


Mesh Module Example

Cartesian Grid Module

- Used to create, edit, and visualize rectilinear grids
- Datasets can have values at cells, corners, and midsides

More Info...

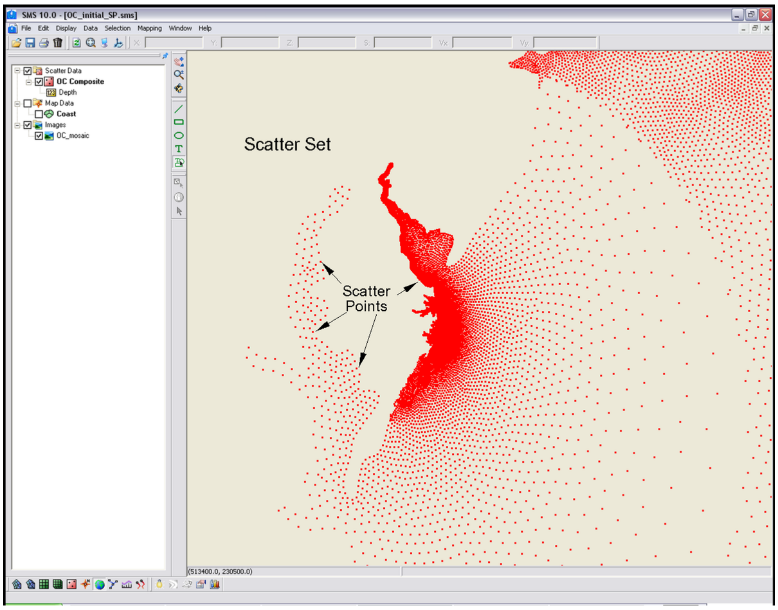


Cartesian Grid Example

Scatter Module

- Used to create, edit, and visualize triangulated irregular networks
- DEMs can be read in and converted to TINs
- Filter scatter sets to eliminate redundant data
- Datasets can be interpolated to other modules (meshes, grids, etc)

More Info...

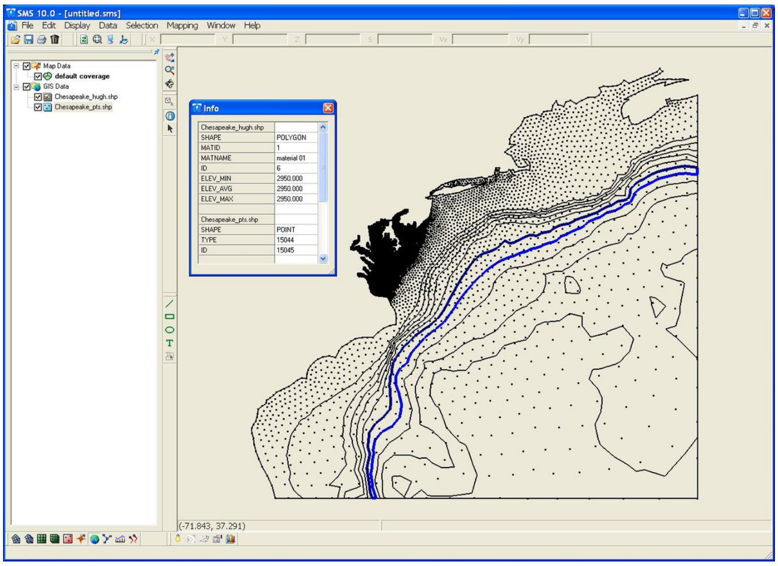


Scatter Set Example

GIS Module

- Open and visualize GIS data
- Supports ESRI and MapInfo formats
- Uses Mapobjects for ESRI files if available to use ArcGIS visualization options
- GIS data can be converted to feature data (map module)

More Info...

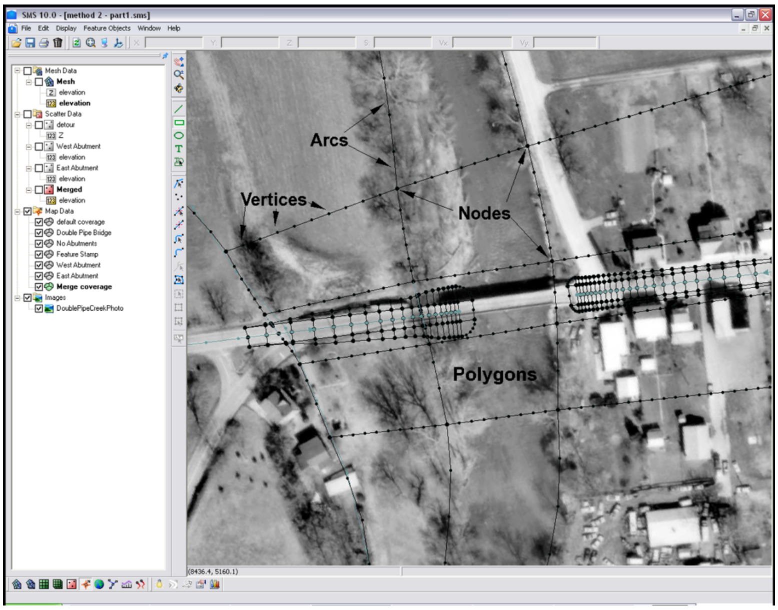


GIS Data in SMS

Map Module/Conceptual Models

- Create and edit GIS like data
- Used to create conceptual models as well as data for other purposes
- Conceptual model is a geometry (mesh/grid) independent representation of the numeric model domain and/or boundary conditions
- Conceptual models can be converted to model geometry and boundary conditions
- Conceptual model makes it easier to create, edit, and alter models

More Info...

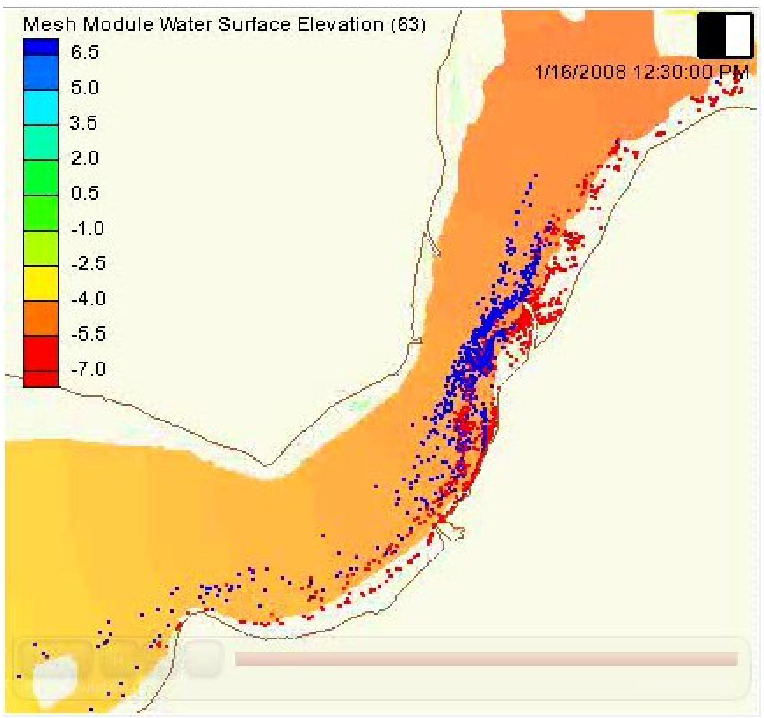


Map Module Example

Particle Module

- Visualize particle/path data
- Supports PTM module which computes particle positions through time based upon hydrodynamics and wave effects

More Info...



PTM Particle Trace

Models

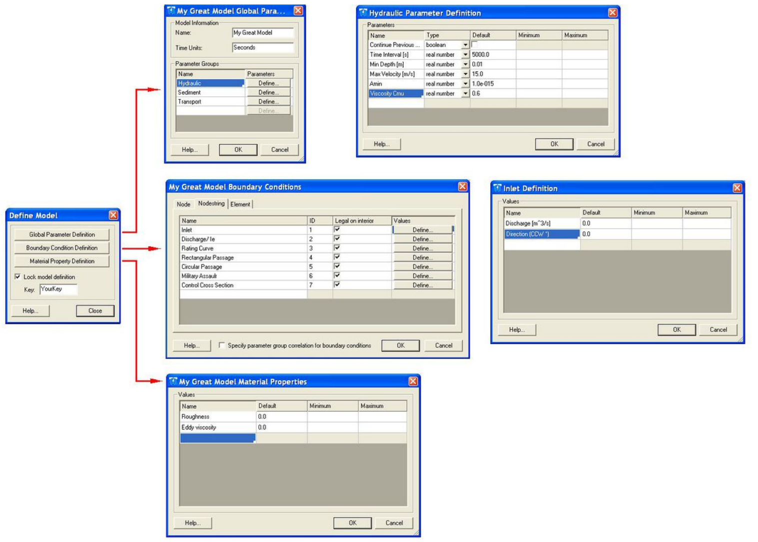
- SMS is a graphical interface that supports many numeric models
- The models were developed by government or private entities
- Hydrodynamic models compute water surface elevations and velocities
- Wave models compute wave characteristics
- Genesis is a shoreline model that predicts how the coastline will move based upon long term wave information
- PTM tracks particle positions through time based upon hydrodynamics and wave effects

More Info...

Generic Model Interface

- Allows creation of a user defined mesh module interface to use SMS with a model not natively supported
- User defines available model parameters and boundary condition options
- User defined interface can be used to build models
- User data is exported into ASCII data that can be read as input for a numeric model

More Info...



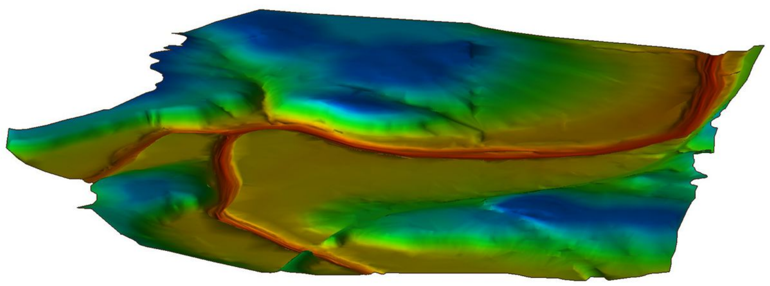
Generic Model Definition and use

Visualization Tools

Contours

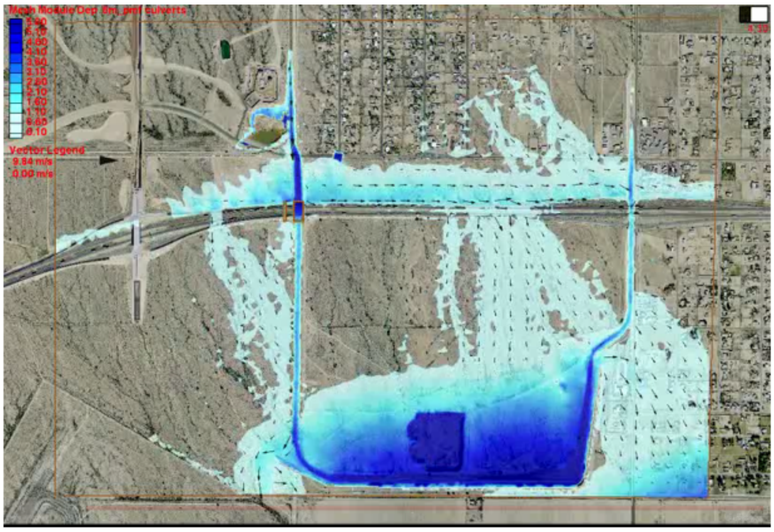
- Visualize scalar datasets
- Linear, color filled or both at the same time
- Variable level of transparency
- Full control of ranges and colors
- Precision control for labels and legends

More Info...



3D view of color filled contours

Vectors

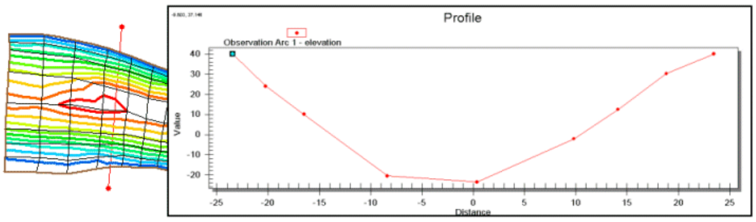


Vectors display

- Visualize vector datasets as arrows
- Constant size or vary by magnitude
- Show just a range of magnitudes
- Color by magnitude

More Info...

Plots



A plot showing a cross-section view of a river channel

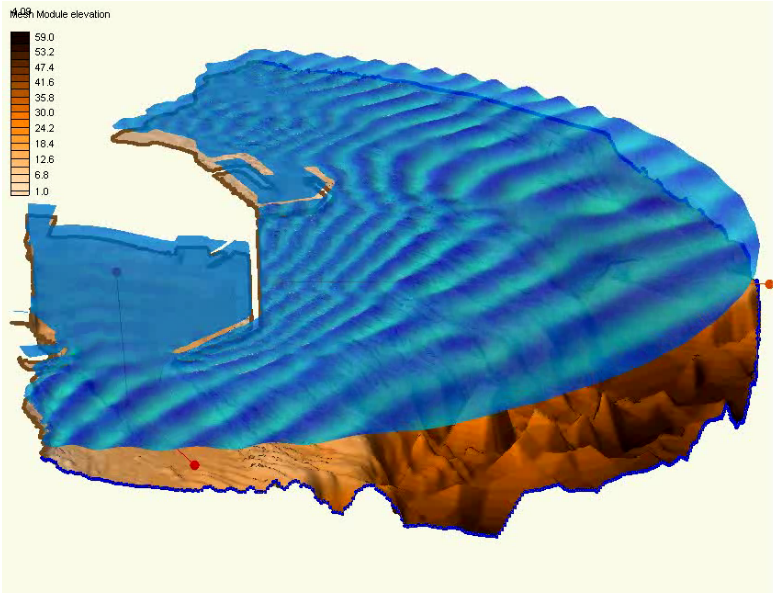
- 2D plots to visualize results and compare to measured values
- Profile plots view scalar data along an arc
- Time-series plots view scalar, vector, or flux (flowrate) data at a point or across an arc
- Several kinds of plots can be used to compare model results with measured data

More Info...

Functional Surfaces

- Surface with elevation based upon scalar dataset values
- Very useful for wave models and models with large change in water surface elevation
- Elevations can be exaggerated to better visualize dataset variations
- Surfaces can have a solid color or use color filled contours
- Transparency can be used to allow see through surfaces

More Info...



Wave functional surface

Animations

- Several types of AVI animations (film loops) can be generated by SMS
- Transient data animation shows model changes through time (contours, vectors, etc)
- Flow trace uses vector data to generate flow paths through the geometry
- Drogue plots use user specified starting locations and show how the particles would flow through a vector field
- Multiple view animations show the data while transitioning between different views


Flow trace animation

- Plot window animations show plots changing through time

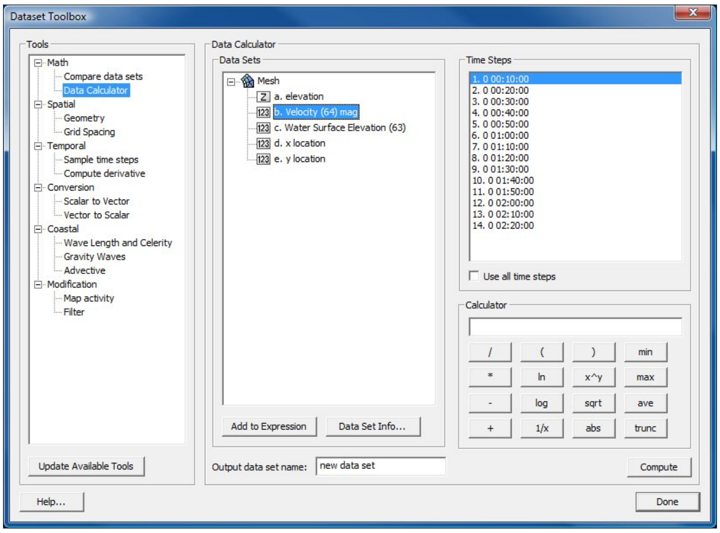
More Info...

Data Tools

Data Calculator

- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

More Info...

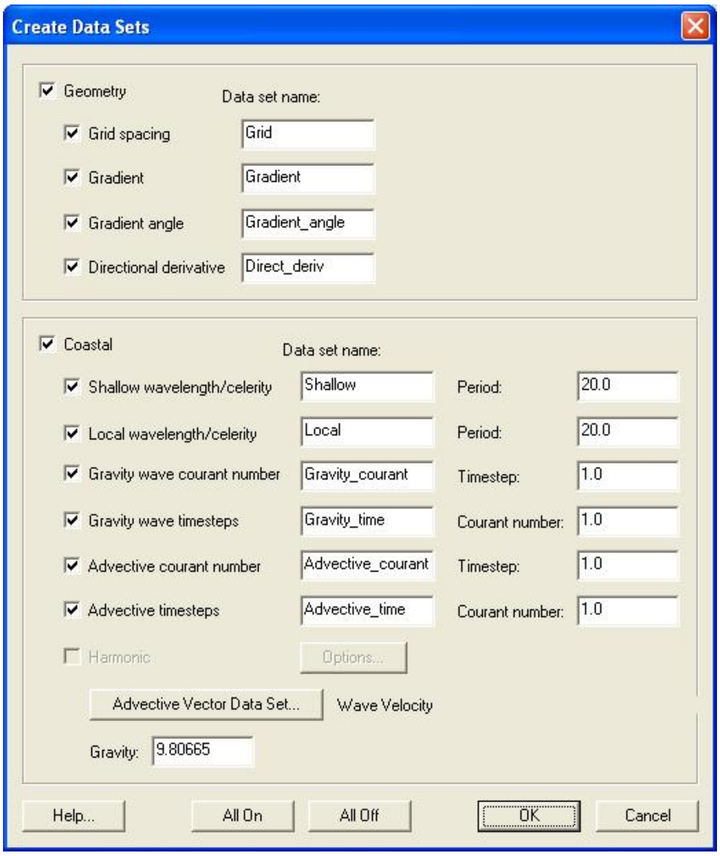


Data Calculator

Create Datasets

- Automatically generate commonly used datasets

More Info...

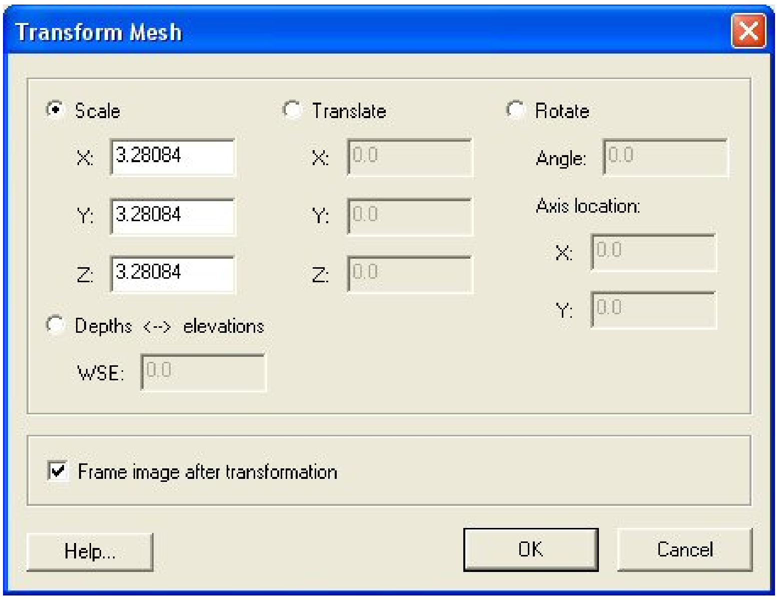


Create Datasets

Data Transform

- Data can be scaled, translated, rotated
- Depths/Elevations can be converted back and forth

More Info...

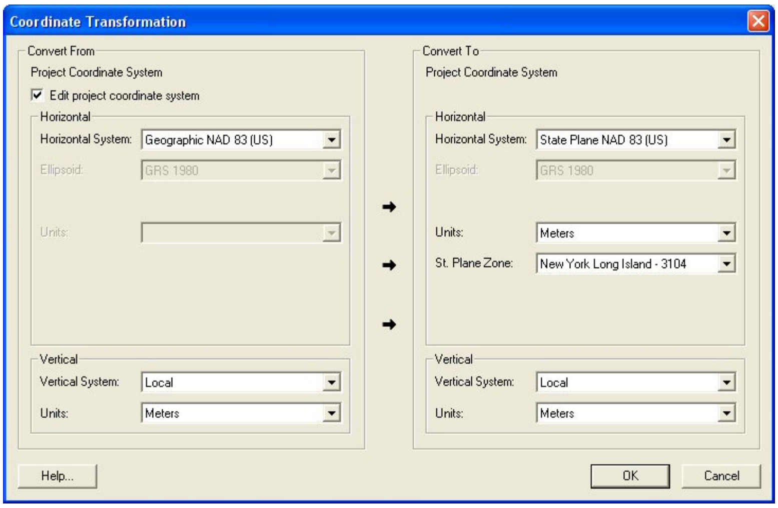


Transform Data

Coordinate Systems

- Associate a coordinate system with your data
- Convert data from one coordinate system to another
- Coordinate systems include Geographic, UTM, and State Plane coordinate systems

More Info...

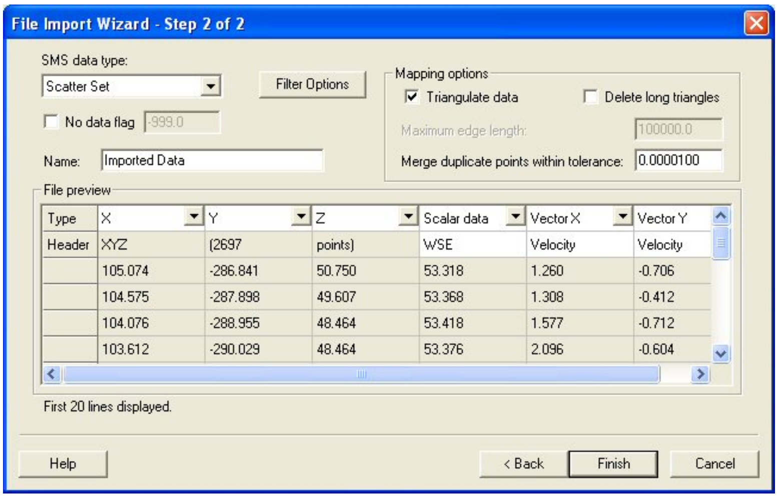


Coordinate Conversions

Import Wizard

- Read columnar ASCII data into SMS
- Columns can be fixed width or delimited by specific characters
- Data can be read as mesh, scatter, or map data

More Info...



Text file read in through import wizard

Miscellaneous Tools

Image Support

- Multiple images can be read/viewed at the same time
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Images can be geo-referenced to view images along with other data
- Images can be draped over mesh or scatter data
- Image pyramids can be created which improves viewing at various zoom levels

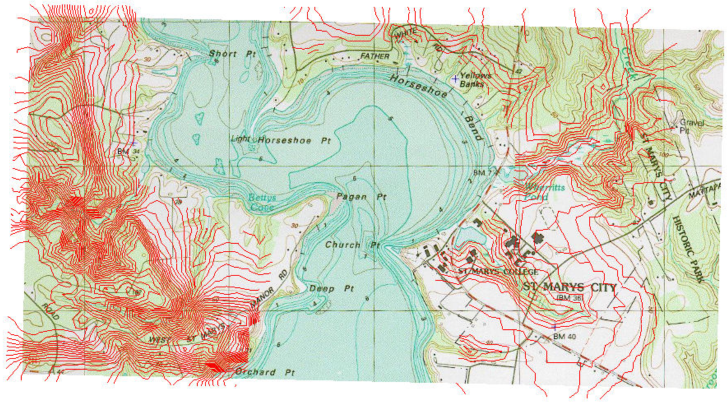
More Info...

A color aerial photo under a semi-transparent topographic map

CAD Support

- AutoCAD DXF and DWG files can be read into SMS (support of DGN format is under development)
- Supports up to AutoCAD version 2007
- CAD data is displayed in 3D
- CAD data can be converted to map or scatter data

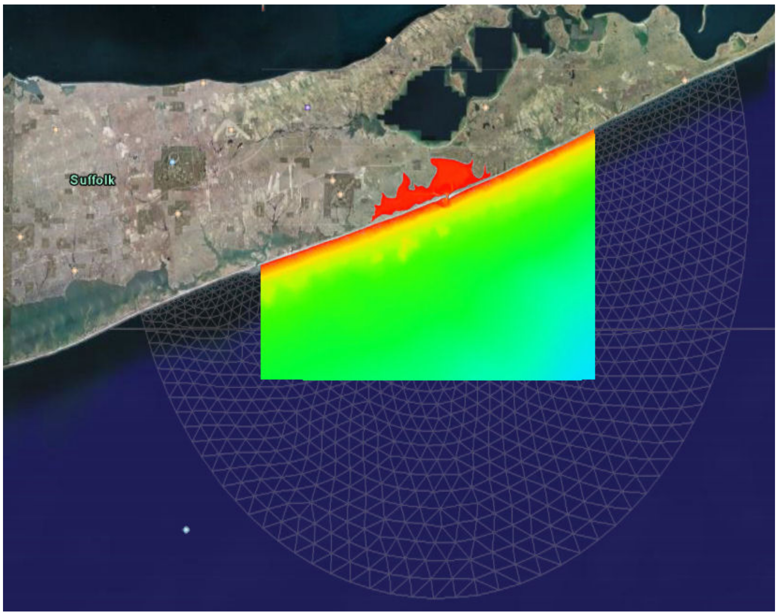
More Info...



Using CAD data in SMS

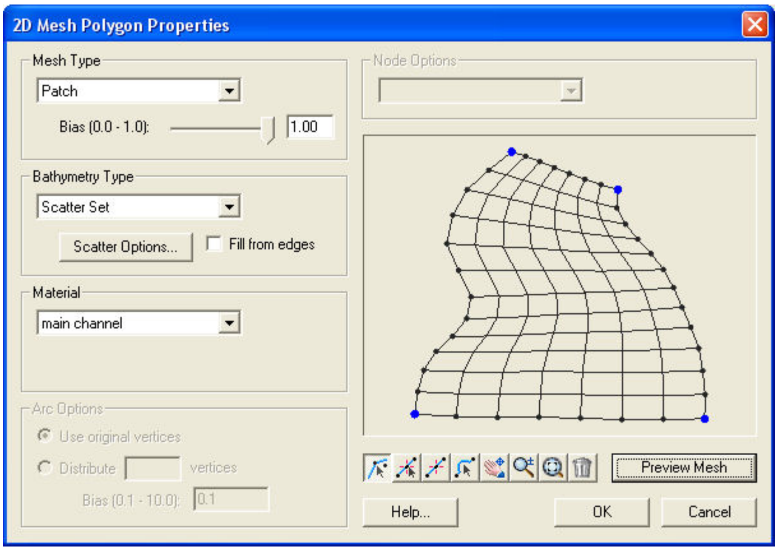
Export Options

- Graphics window can be copied to the clipboard
- Current view can be exported in KML format for visualization in Google Earth



Model Data Visualized in Google Earth

Meshing Options

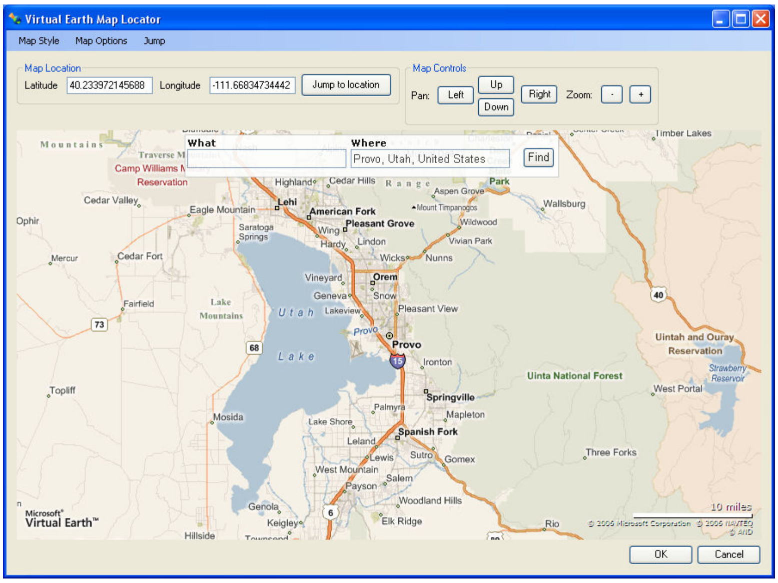


2D Mesh Polygon Properties Dialog

- Generating a quality finite element mesh is central to using many SMS models
- Conceptual models make generating meshes easier
- Polygons can use a variety of meshing options to generate triangular or quadrilateral elements
- Polygons can be assigned bathymetry and material information that will be transferred with the mesh
- Scalar paving density generates elements with sizes based upon a size dataset allowing for smooth transitions and a large range of element sizes and is particularly useful for coastal and wave models.
- Datasets for scalar paving density can be user defined or generated using the data calculator, the create datasets command, or LTEA (linear truncation error analysis) (ADCIRC)

More Info...

Import from Web



Virtual Earth Map Locator

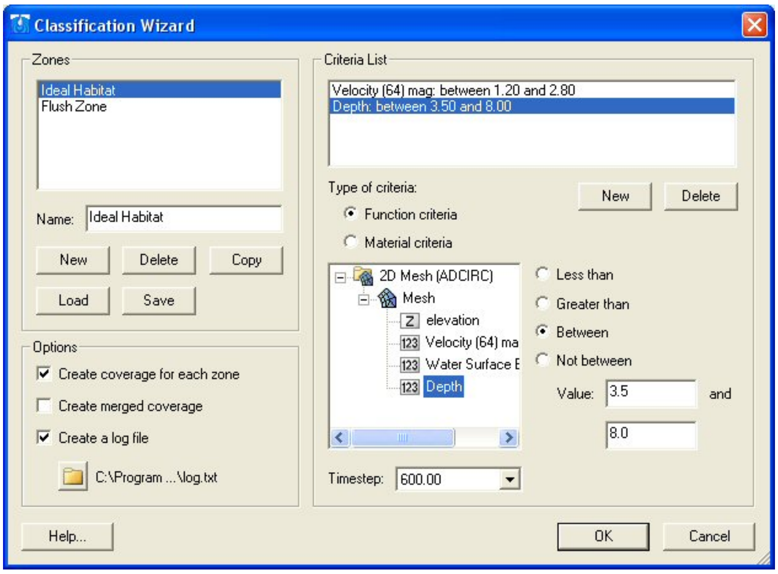
- Easy to use navigation tool allows user to choose model location
- Image data is downloaded from USGS terraserver
- Image options include aerial photos, topographic charts, and urban (higher resolution color)

More Info...

Zonal Classification

- Generate a map coverage identifying areas that meet specific requirements
- Requirements can be based upon dataset values such as less than a specific value or based upon materials in an area property coverage

More Info...



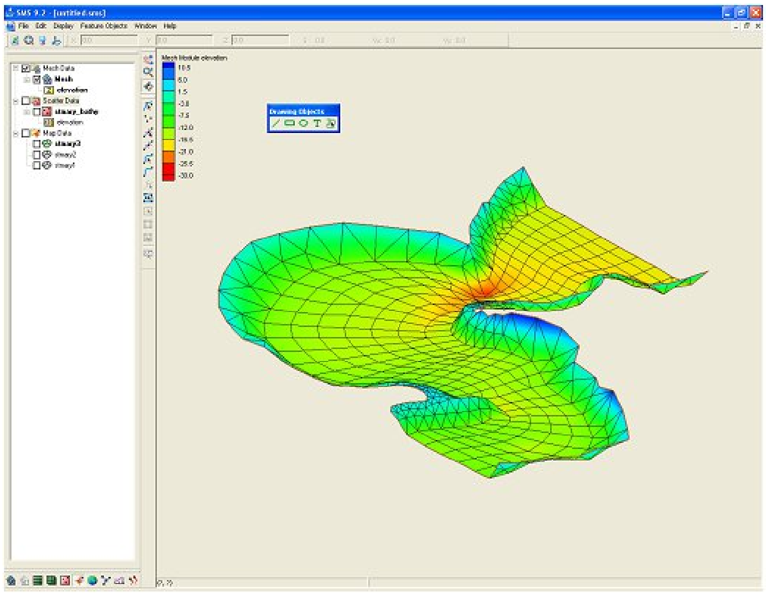
Zonal Classification

SMS Screen

The SMS screen is divided into six main sections:

1. Main Graphics Window
2. Project Explorer – this may also be referred to as the Tree Window
3. Toolbars
4. Edit Window
5. Menu Bar
6. Status Bars

Normally the main graphics window fills the majority of the screen; however, plot windows can also be opened to display 2D plots of various data.


The SMS screen

What's New in SMS 11.1

Release Notes

SMS 11.1.1 – Beta Release December 11, 2012

This is a bugfix release for SMS 11.1.0

You must uninstall the previous beta versions of SMS 11.1 before.

Bugfixes

The following bugs have been fixed in this version.

1. Resolved issues related to interpolating from rasters or raster sets to scatter sets.
2. Resolved issue related to the display of contours on a raster (or raster set) as a surface. Scalar values were being offset by half a raster cell.
3. Added a display state for each raster when the raster is saved to an SMS project so that they are not turned on when the project is loaded.
4. If a user tries to process boundary triangles when the display projection is not the same projection used by the scatter set will immediately tell the user that the display projection must be changed.
5. The order of rasters in the list will now control the drawing order. Dragging a raster to a lower position in the raster set will hide that raster behind rasters above it in the list.
6. Messages related to reading incomplete AdH solutions have been modified to reduce the confusion. Redundant messages have been removed. AdH solution folders are now preserved when a project is saved.
7. Fixed issue with converted selected map (feature) objects to a scatter set.

SMS 11.1.0 - Beta release October 8, 2012

SMS 11.1 is a general feature upgrade from SMS 11.0. Standard installation is in a different directory than that used by SMS 11.0 and therefore, both versions can be installed on the same machine.

Licenses for SMS 11.0 which are up to date on their maintenance enable the same corresponding modules in SMS 11.1.

While the version is in beta release, the executable will expire after 90 days. This is because we anticipate some revisions and corrections and we want to encourage users to work with the most stable version possible.

Automatic bug fix versions are posted nightly on the download site. These are not official releases, but can be used if bugs are encountered in the beta release and the support department indicates these issues are resolved but not officially posted yet.

General Features

Dynamic Images

- SMS can now link to dynamic image sources on the web so that the background image is updated as user pans/zooms around the domain. These images can be converted to static images for speed or if internet access may be lost.

New Online Image Sources

- Recently, the terraserver web service that had been available in SMS was discontinued. SMS now accesses a number of new web sources for image and/or elevation data using the previous web source tools.

Project on the fly

- All data in the SMS instance is now projected to a user specified projection for display. This makes working with data in various projections much easier and less prone to round-off or conversion errors.

Dataset Toolbox

- Angle convention conversion for scalar datasets. For more information, see Dataset Toolbox.

Open as... command

- General data formats such as raster, vector and image files are not always supported as native in SMS. This new command allows SMS to use the Global Mapper library to convert the data in these files to be read in to SMS.

Import Wizard column association

- The import wizard can now associate Northing/Easting headers with Y/X.

Annotations

- In addition to previous annotation capabilities, users can setup annotation layers so they are only visible during user specified time intervals. This would apply to film loops and stepping through time steps. See Annotations for more information.

Module Features

GIS

Convert GIS data to Generic model coverage

- Users can now bring in GIS data (shape file or mif/mid) and convert this to generic model node or arc attributes. For more information, see GIS to Feature Objects Wizard.
- Read in a boundary condition file into the GIS module. For more information, see PTM Boundary Condition File.

Map Module

- Select all arcs connected sequentially to define a polygon. Can be used to detect poor connectivity in a conceptual model. For more information, see Select Connected Arcs.
- Split arcs based on user specified criterion. For more information, see Split Feature Arcs Utility.
- Convert Map feature arc and point attributes (Generic 2D mesh coverage) to mesh nodestring and node attributes. For more information, see Generic Mesh Coverage.
- The Select/Delete Data function (Feature Objects menu) now gives the user the ability to set how points that lie on the selection polygon will be treated. For more information, see Select/Delete Data....
- Map data can be imported/exported in MIF/MID format. For more information, see Export map data in MIF/MID format.
- Export map data in shape format. For more information, see Export Map Data in Shape Format.
- Map files have been optimized to load faster.
- Converting a area property coverage to a 2D grid has been optimized to convert faster.

Mesh Module

- Nodestrings now contain an id. For more information, see 2D Mesh Nodestrings Menu.
- A subset of the mesh can be selected and edited on its own. For more information, see Editing 2D Meshes.
- Meshes can be imported/exported in MIF/MID format. For more information, see Export Meshes in MIF/MID Format.
- An option now exists for mesh to scatter conversions where only selected mesh nodes are converted.
- Automatic zoom was removed when generating a mesh from a map coverage.
- Reduce nodal connections. For more information, see Reduce Nodal Connectivity.
- Global mesh renumbering done using the Cuthill-Mckee renumbering scheme replaces nodestring based renumbering.
- Mesh quality scatter set creation. For more information, see 2D Mesh Module Menus.
- ARR plot creation. For more information, see ARR Mesh Quality Assessment Plot.

Particle Module

- Extract particle subset – Export a portion of the times and/or particles to a new file to reduce file sizes of particle solutions.

Raster

- New raster set tree item folder in the project explorer. For more information, see Raster Module.
- New "Options" dialog in the raster set project explorer. This is used specify if raster values are/aren't elevations. For more information, see Raster Functionalities.
- Ability to convert part of a raster to a scatter set (TIN).

Scatter Module

- Ability to autogenerate TIN breaklines following elevation values. For more information, see Generate Contour Breaklines.
- New option to use Laplacian interpolation when interpolated to a Cartesian Grid. For more information, see Laplacian Interpolation.
- Merging scatter sets will now show a report of the number of vertices, triangles, breaklines before and after the merge. For more information, see Data Module Commands.
- New scatter filter using VTK Decimate Pro algorithm. For more information, see Scatter Filter.

Cartesian Grid Module

- We now have a floating projection option. For more information, see Cartesian Grid Coordinates.

Model Features

AdH

- Support for version sediment and constituent transport.
- Preservation of "unsupported" or "advanced" cards in an "Advanced" tab in model control
- Support for weir structures.
- Support for PC LVL card (output control).
- New "Sediment Library Control..." dialog.

ADCIRC

- Support for version 50.0. New distribution of the model.
- Support for running ADCIRC in parallel (PADCIRC) on windows environment.
- Support for the new NWS 19 option

CMS-FLOW

- Support for version 4.5 of the CMS. This includes both CMS-Flow and CMS-Wave in a single executable.
- Complete overhaul of the CMS-Flow model parameters.
- New parameters to the "Sediment" tab. For more information, see CMS-Flow Model Control.
  - Calculate morphology change during ramp period
  - Avalanching
  - Made the D50 Dataset optional
  - Added a new dataset called Sediment Standard Deviation (mm)
- Changes to the wave model control tab
- Added new Harmonic WSE-forcing boundary condition. This is located in the CMS-Flow Model Control options under "Tidal" tab. For more information, see CMS-Flow Model Control.
- Added Longitude dataset. You will see this option in the CMS-Flow Model Control options under "Flow" tab. For more information, see CMS-Flow Model Control.
- Added 29 new Tidal Constituents bringing the total to 37. For more information, see CMS-Flow Model Control.
- Added Grid modification flag. For more information, see CMS-Flow Grid Modification Flag.
- Added ability to read netCDF files for visualization.
- Added ability to read fleet wind data. For more information, see Fleet Wind Files.
- New save point coverage and file support. For more information, see CMS-Flow/Save Points.
- Ability to extract levee/structure height from an unstructured (ADCIRC) grid.

CMS-Wave

- Support for updated version
- We now write the *.std file according to the new format.
- Users can associate times/dates with the simulation.
- We now support full plane simulations.
- Ability to create input spectral locations in spectral coverage. For more information, see CMS-Wave Spectral Coverage.
- Support of the new permeable structures.
- Support spatially varied wind input fields.
- Support for 8 (yymmddhh) or 12 (yyhhddhhmmss) date fields for temporally changing spectra.

CSHORE

- We now can interface with CSHORE. For more information, see CSHORE.

CStorm

- Exporting ADCIRC/STWAVE & ESMF
- We now write the STWAVE and ADCIRC files into a single directory along with the ESMF mapping information.
- Allow users to specify file links instead of data ojects for ice and currents.

Generic Model Interface

- Converter utility provided to model developers to facilitate migration from previous template to SMS 11 template.
- Ability to only show material options for active material group.

PTM

- You can now read in PTM trap files. For more information, see PTM Trap File.
- Ability to use an ADCIRC ascii fort.45 files for hydrodynamics.
- Improved smoothing of display of 3D solution of fence diagrams.
- Support to read in BC file.
- Improved interface with CMS-Flow.

STWAVE

- You can now run stwave in parallel if you have multiple processors. For more information, see STWAVE.

TUFLOW

- Can specify which TUFLOW executable (double or single precision as well as 32 and 64 bit) to use for each simulation.
- Improved model check for boundary conditions for specified event.
- Add ability to specify a clip region.
- Convert GIS rainfall data to TUFLOW boundary conditions
- Provide option for GIS check/outputs to be shapefiles
- Support mass balance corrector toggle
- Support new Source over Area commands
- Support storage reduction feature
- Support variable z shape restore/repeat

- Rearrange grid options dialog
- Improvements to multiple file import dialog (open TCF file)
- Support for customary units option

Features still under development but not available for public use

These are experimental tasks. If you are interested in them, please send an inquiry to tech support (support@aquaveo.com).

Curvilinear (Orthogonal) Grid Generation

- Users can specify in the simulation's model control how many layers there are for their curvilinear grids.
- Users can now use a masking coverage to create more precise curvilinear grids. For more information, see LTFATE.
- New feature for relaxing curvilinear grids using predictor/corrector algorithm.
- User can now display 3D solutions mapped onto fence diagrams cut through the domain.
- Support for the LTFATE/CH3D model parameters and file formats.
  - Curvilinear Grid LTFATE supports Tidal and Inflow boundary conditions. For more information, see LTFATE.
  - Added support for Sediment river boundary conditions. For more information, see LTFATE.
  - Added support for wind.inp file. For more information, see LTFATE.
  - Support for SEDZL-J sediment transport parameters.
- Support for the EFDC model parameters and file formats

VTK Dataset Interpolation

- Can interpolate from a VTK dataset to mesh2d, cgrid, scatter, vtk mesh or curvilinear geometric object. For more information, see Interpolation (VTK).

1.1. General Information

Bugfixes SMS

Please see the SMS Intermediate Release Bugfixes page for the version of SMS you are interested in:

SMS 11.1 Release Bugfixes

SMS 11.1.1 - Beta Release January 17, 2013

This is a bugfix release for SMS 11.1.1

1. Fixed an issue with AdH Model losing data.
2. Fixed issue related to TUFLOW 1D causing a crash.
3. Fixed issues related to AdH Sediment Bed Layers tab.
4. Resolved an issue causing processing boundary triangles to miss some triangles.
5. Fixed a crash caused from reading in a bad .sol file type.
6. Fixed a crash related to AdH Nodal Boundary conditions.
7. Fixed a crash caused by saving out BOUSS2D damping information.
8. Fixed a crash related to merging scatter sets with delete old sets option.
9. Resolved a tabbing order issue in the AdH model interface.
10. Resolved an issue with dragging certain coverages into the TUFLOW simulations.
11. If a user tries to read in an incomplete .sol file type they will now be notified that it is incomplete.
12. Resolved an issue where SMS was not writing all necessary .hot information out when saving.

SMS 11.1.1 – Beta Release December 11, 2012

This is a bugfix release for SMS 11.1.0

1. Resolved issues related to interpolating from rasters or raster sets to scatter sets.
2. Resolved issue related to the display of contours on a raster (or raster set) as a surface. Scalar values were being offset by half a raster cell.
3. Added a display state for each raster when the raster is saved to an SMS project so that they are not turned on when the project is loaded.
4. If a user tries to process boundary triangles when the display projection is not the same projection used by the scatter set will immediately tell the user that the display projection must be changed.
5. The order of rasters in the list will now control the drawing order. Dragging a raster to a lower position in the raster set will hide that raster behind rasters above it in the list.
6. Messages related to reading incomplete AdH solutions have been modified to reduce the confusion. Redundant messages have been removed. AdH solution folders are now preserved when a project is saved.
7. Fixed issue with converted selected map (feature) objects to a scatter set.

SMS 11.0 Release Bugfixes

The following bugs have been fixed in this version.

1. Error in converting a stamped coverage to a scatter set was resolved
2. When loading a generic mesh in a *.2dm file, with a template that does not correspond to the current model definition, SMS could crash. This problem was resolved to allow for preservation of the grid.
3. In some situations, when generating a drogue plot or particle trace animation, some of the particles were not getting the new time step data. This has been resolved.
4. Fixed an error related to interpolation to nodes that are “inactive”. All nodes now get interpolated data regardless of the state of the connected elements.
5. Calibration plots using the active data set did not update when changing to a completely different scatter set. This has been resolved.
6. Fixed a drawing order that caused annotation items to appear on top of the film loop clock. The clock now appears on top.
7. Additional comments were added to the TUFLOW Flow Constriction dialog to make the units and coefficients more understandable.
8. Bug related to reading in a large RMA2 solution was resolved.
9. When SMS writes GIS data (or a coverage) as a *.mif/*.mid pair, an extra “bound” record was being written. This has been corrected.
10. Calibration targets are now not drawn when the observation point falls in an inactive cell/element.
11. When framing the display, SMS now includes active grid frames.
12. Fixed erase behind labels to include the film loop clock.
13. When creating datasets in the dataset toolbox, if all arguments to an operation come from a single folder, the result will now be placed in the same folder.
14. Fixed a situation where the screen froze trying to display contour labels.

SMS 11.0.11 – Built October 5, 2012

1. Several internal memory leaks were corrected.
2. How SMS saves the input wind file for the “Holland Symmetrical” hurricane option for wind in ADCIRC were corrected.
3. An error related to material sets in TUFLOW was corrected. SMS would lose track of which material set was assigned to a simulation when a project was saved and then read back in.
4. In some cases it appeared that SMS was deleting data sets from the Project Explorer. In reality, there were just some empty folders that were left in a simulation.
5. Clarification was made for the “Override Z values” option for TUFLOW in both the dialog and the documention.
6. In the dataset toolbox, when SMS creates a new data set, the activity was being computed from the resultant nodal activity. This caused single cell/element islands that were inactive to become active. This was changed to use the element activity of the input data sets to compute the resulting element activity.
7. The ordering of external files in the geometry component file (tgc) is now controlled by the order of the tree items to allow the user to control this explicity.
8. A crash when loading ADCIRC grid files was resolved. Some files had “Unexpected End of File” messages, and then crashed. They now read in correctly as they did with previous versions.
9. Error in the TUFLOW interface to “Force cell z to at or below the node z” was resolved. The option is now available without activating the “Set Channel Invert” elevation.
10. Inconsistent display of rasters corrected.
11. Error in the SMS steering of CMS-Wave and CMS-Flow was corrected to save the correct eta files.
12. A file pointer to a shape file was not released when SMS opens the file. This has been corrected.

13. A issue with user defined palettes was corrected. The first palette was always active when reloading a project. Now SMS looks for a named palette that was active when the project was saved. If that palette exists, it is made current.
14. SMS has been known to save an illegal material with an ID of zero when saving TUFLOW projects. This should not happen any more.
15. SMS allows two TUFLOW boundary condition coverages. They cannot be identical or TUFLOW will crash.
16. Fixed an issue of editing BC curves associated with arcs in a TUFOW coverage. Changing the curve used to change it for other copies. The distinction of multiple curves is now maintained.
17. Fixed a bug with how WAM grids read in native format are loaded into SMS.
18. Fixed output of GCL strings for RMA4 to not include the midside nodes.

SMS 11.0.10 – Built September 10, 2012

1. Additional documentation was added to the help manual on the AdH model control options.
2. A memory overflow error when triangulating is detected so that SMS does not crash when triangulating very large scatter sets. Several possible solutions to get the data loaded are suggested. A future enhancement will reduce memory requirements for triangulation. This also applies when merging two large scatter sets that cause SMS to try to retriangulate.
3. Fixed problem when loading STWAVE output files and converting the data sets to scalars. The conversion was creating scalar data sets with “0.0” values.
4. Fixed an issue with graphics when dragging an arc in the Map Module. During the drag, a phantom arc would appear offset from the arc that was being dragged.
5. Fixed a problem when forcing breaklines into a TIN that resulted in flat or degenerate triangles. No degenerate triangles should be created now.
6. A problem with RMA4 giving bad values at midside nodes was isolated and reported to the TABS team at ERDC. No update is expected.
7. A problem was resolved in the “Holland Symmetrical” hurricane option (NWS = 8).
8. Resolved an issue with calibration plots when multiple solutions exist.
9. Fixed spacing in the model wrapper when running AdH to match what is output in a command prompt. This makes the output more readable.
10. Fixed a crash when creating a telescoping grid for CMS-Flow.
11. Resolved an issue when reading a TUFLOW solution with overlapping elements at the confluence of two 1D tributaries. SMS deletes the longer (skinnier) triangles that were overlapping, but still reads the data sets for visualization. When these cross sections are properly constructed, there should not be overlapping triangles.
12. Made change so that when user tries to open the HURDAT database, SMS switches to geographic coordinates to support the data if possible.
13. Fixed crash related to SMS trying to switch into PBL model even though the interface is not licensed.
14. Resolved a situation which caused SMS to freeze when dragging a coverage to another location in the project explorer (in Map Module or in a TUFLOW simulation).
15. Clarified the management of linear –vs- quadratic elements in the mesh module. If both types are permissible, SMS now asks the user to specify the desired type. The default is set to quadratic if quadratic elements exist of when working with TABS or FESWMS. Otherwise, the default is linear.
16. Fixed a crash when loading multiple map files into SMS.
17. Added check to verify validity of a coastline file before reading to prevent crash.
18. Fixed a situation where SMS assigns the wrong material type to polygons created during a conversion from a mesh.
19. Added support for “TD Card” (the global time step) when converting SMS 10.1 generic model templates to SMS 11.0 format.

20. Added a warning to let users of the generic model interface be informed that just reading a 10.1 template will not allow use of an 11.0 generic model version.
21. Fixed conversion of air density for FESWMS when switching from customary units to SI.
22. Implemented a change to maintain the material set assignments for TUFLOW simulations for an SMS project when saving and reopening the project.
23. Corrected a linkage problem with scatter data sets being lost when deleting other data sets.
24. Clarified the documentation and prompts when using the “override Z values” options in the TUFLOW interface. The model supports several options that were causing confusion among users.
25. Added a right click command in the 1D-2D connections coverage properties to allow users to select the boundary condition coverage to use with 1D-2D connections in the TUFLOW interface.
26. Fixed a problem with CMS-Flow/CMS-Wave steering that was preventing the “eta” or surge file from being passed to the wave model after a flow simulation.
27. Fixed a problem with the names of rasters being reset to default value after saving.

SMS 11.0.9 – Built August 6, 2012

1. Fixed an issue in the CMS-Flow model wrapper to ensure the run time would update.
2. Fixed a problem with the names of rasters being reset to default value after saving and reloading an SMS project.
3. Fixed problem with visualization mode to support quadratic elements. This allows switching between visualization mode and TABS without problems.
4. Fixed error in computing flux across an observation arc. Documented that insufficient resolution, either by saving only TUFLOW corners, or by unrepresented AdH refinement can cause errors in computed flux.
5. Fixed bug with bad air mass density when converting from customary units to SI and back.
6. Fixed bug with node labels that stayed visible when nodes were turned off.
7. Fixed display of node numbers so that they do not overlap nodal elevations when both are turned on.
8. Fixed a problem with the name that was written to the *.mp file for CMS-Flow when extracting both WSE and velocities along a boundary cellstring.
9. Fixed a bug when reading in *.2dm file boundary condition on a deleted node string.
10. Fixed issue with the display of vectors after the mesh is turned off.
11. Fixed a crash in the AdH materials property dialog when using sediments.
12. Fixed a tolerance issue when using the Paving function with geographic coordinates and inserting a new mesh into an existing mesh.
13. Fixed crash when trying to open an STWAVE *.sim file as a CMS Wave *.sim file.
14. Fixed a bug with the TUFLOW 2D Model Control switching from a Coverage selector to an edit field.
15. Fixed a crash that occurred sometimes when exporting TUFLOW files.

SMS 11.0.8 – Built July 3, 2012

1. Fixed an error with exporting STWAVE simulation files.
2. Fixed a bug that where SMS was crashing when trying to remesh a portion of a mesh when quadratic elements were used.
3. Fixed an issue where nodestring IDs were not read in correctly.
4. Fixed a problem where some image files did not display until another item was read in.
5. Fixed a problem with the generation of damping cellstrings for BOUSS2D.
6. Corrected an issue first seen in SMS 11.0.4 where internal arcs were not handled correctly when creating a 2D mesh.
7. Allowed negative values for TUFLOW output start times since TUFLOW allows negative times.
8. Fixed an issue with ADH iteration controls not reading/writing correctly.

9. Fixed a crash that sometimes happened when converting GIS data to feature objects.
10. Fixed a problem where the initial contour display was a solid blue rather than specified values.
11. Corrected a problem where TUFLOW material sets were put into folders incorrectly.
12. Fixed an issue where SMS was creating bad triangles when merging scattersets.
13. Dragging of feature nodes now displays connecting lines while dragging.
14. Removed an incorrect model check warning when using extracted boundary conditions with CMS-FLOW.
15. Fixed a crash that sometimes happened when opening two *.tcf files at the same time.
16. Various bugfixes from software crash reports.

SMS 11.0.7 – Built May 31, 2012

1. PTM will no longer show not licensed when run from SMS using a single user lock.
2. Mesh quality not displayed when a mesh is turned off in the project explorer.
3. Datasets are preserved when saving/loading a project file which wasn't always working.
4. When displaying labels on scalar values at mesh nodes, the labels could disappear sometimes. This has been fixed.
5. Fixed a problem where reading certain *.mif/*.mid files crashed SMS.
6. Fixed a problem where SMS was hanging when cleaning/merging arcs.
7. SMS sometimes gave an error message "Unable to write file path for keyword ":NEIGHBOR_FILE" when saving an SMS project.
8. Fixed a problem where SMS would freeze trying to perform zonal classification.
9. The CMS-FLOW folder was reorganized to allow 32 and 64 bit executables to both work.
10. Sometimes SMS was not maintaining the folder structure for loaded XMDF dataset files.
11. Fixed a problem where zonal classification was not working correctly for Cartesian grids.
12. TUFLOW events now prevent commas from being used as they don't work with the TUFLOW file formats.
13. Fixed an issue where right-clicking on coverage and choosing Map→2D Mesh would operate on the active coverage rather than the one clicked on.
14. Fixed a problem where the TUFLOW files for geometry components were written out incorrectly when multiple components exist.
15. Fixed an update problem when using user defined palettes.
16. Fixed a problem where rotating could cause parts of a functional surface to disappear.
17. Fixed a problem where the data calculator was not correctly using inactive cells for CMS-FLOW.
18. Fixed a location where SMS was giving a prompt that the project had changed when it hadn't.
19. Changed SMS to not reframe when performing a map→2D mesh.
20. Fixed a crash when working with user defined tidal constituents in ADCIRC.
21. Fixed an issue where breaklines of a scatterset were being displayed even though the scatterset was turned off in the project explorer.
22. Fixed a problem where the runtime wasn't being updated when running CMS-FLOW.
23. Turned off a bogus model checker when using CGWAVE when using approximated wave conditions.
24. Fixed a series of bugs related to setting up a generic model template.
25. Fixed an issue where SMS was not writing the correct number of frequencies to the STWAVE *.eng file.

SMS 11.0.6 – Built May 4, 2012

1. Fixed a bug that was introduced in SMS 11.0.5 where SMS would hang when trying to build polygons.
2. Included a file needed to check for AVI codecs to make sure they would work with SMS. This file is only needed for versions starting with SMS 11.0.5.
3. Fixed a minor issue where the welcome dialog reported that the SMS was a beta version.
4. Miscellaneous changes based upon crash logs sent from the software (bugtrap).

SMS 11.0.5 – Built April 25, 2012

1. Fixed a bug that caused SMS to freeze when using a large duplicate node tolerance.
2. Corrected the executable used by the BOUSS2D “1D Runup and overtopping calculator.”
3. Fixed a problem where framing and switching to plan view could change the display of elements and nodes.
4. Tightened the tolerance used when loading RMA2 files to prevent curves from losing information when loading.
5. Fixed a problem that sometimes made the generic mesh interface hang.
6. Fixed a problem where the specified image projection was not written out causing the need to respecify the projection everytime the project was loaded.
7. Fixed a problem where portion of scatter set not displayed correctly when not filling above/below contour range.
8. SMS sometimes fouled up a polygon or crashed when moving nodes in the mesh attributes dialog (polygon attribute in mesh coverage).
9. Corrected an issue where the drogue times in the log file were not reported correctly.
10. Miscellaneous changes based upon crash logs sent from the software (bugtrap).

SMS 11.0.4 – Built March 3, 2012

1. Fixed a problem where sometimes a plot didn’t generate correctly for arcs without interior vertices
2. SMS now closes STWAVE model files after running the model.
3. Fixed a problem that could occur when closing spectral energy plot and then reopening the plot.
4. Fixed an issue with the mesh quality min angle not working correctly.
5. Fixed an issue where the correct tidal constituents was not being displayed when using the Harmonic Analysis option with ADCIRC.
6. Fixed a problem with default values for nodestrings when using the generic mesh model.
7. Fixed a problem where the same coverage name in different folders couldn’t be differentiated when using a coverage to display vectors.
8. Fixed a problem where AdH Iteration control parameters were not being saved.
9. Fixed a problem with reprojecting a Cartesian grid incorrectly reporting the minimum data value.
10. Fixed a problem with curve BC data when using the generic mesh model.
11. Fixed a problem where SMS was freezing when converting scatter breaklines to map.
12. STWAVE half-plane model only supports spectral energy at 5 degree direction bins. SMS was allowing users to specify a different angle which led to problems when running the model.
13. SMS was not handling the float/curve option for the generic mesh model correctly.
14. Fixed a problem where SMS gave a projection error message when loading an image even though the projection was valid.
15. Fixed a crash when trying to paste data into the cross-section attributes dialog.
16. Fixed a crash that could sometimes happen when redistributing vertices inside the polygon attributes dialog.
17. Fixed an issue with losing BC information specified using the generic model interface.
18. Fixed a problem where the spectral index wasn’t being saved correctly.
19. Fixed a problem in how STWAVE with breaking datasets when using XMDF dataset output.
20. Fixed an issue where SMS incorrectly told the user the STWAVE data had changed after saving a file.

21. Made it so that you can control which raster is the active raster.
22. Fixed a problem with zoom to raster item.
23. Made it so SMS would switch modules if you clicked the root raster item.
24. Add raster module to right-click switch modules in empty area of project explorer.
25. Fixed a problem where SMS could freeze when trying to build polygons.
26. Fixed an incorrect model check when working with the CGWAVE model.
27. Fixed a problem with launching generic mesh models.
28. Fixed a crash that could occur when specify custom tidal constituents for ADCIRC.
29. Fixed a crash that could sometimes happen when using functional surfaces and moving windows.
30. Made it so SMS correctly used the projection associated with a scatterset when reading.
31. Corrected an issue where datasets created in the dataset calculator could end up in the hotstarts datasets when using AdH.
32. Fixed a problem where the meshing preview in the polygon attributes dialog for mesh coverages could mess up the polygon's arcs.
33. Fixed a crash with PTM and trap files.
34. Fixed a problem where an error message similar to "No dataset specified for interpolation" when using observation plots with scattersets.
35. Fixed a crash that could sometimes happen when specifying global parameters for a generic mesh model.
36. Fixed an error where SMS gave a generic error message when trying to run STWAVE that did not identify the problem.
37. Fixed a problem where incorrect model setup for WAM could lead to a crash when trying to run the model.
38. Fixed a problem where contours were not being displayed for elements that were wet but being treated as dry.

SMS 11.0.3 – Built Jan 27, 2012

1. Fixed a problem where the name of a spectral grid for CMS-Wave was not preserved.
2. Fixed a problem where grid frames could "disappear" behind other data and not select correctly.
3. Fixed a crash that could happen when trying to convert a mesh to a map.
4. Changed SMS to allow small damping cellstrings for the BOUSS2D model.
5. Fixed a problem with generating cellstrings for BOUSS2D where no cellstring was created on the right side of the grid.
6. Fixed a problem with the generic model interface where boundary condition values were lost after opening and saving.
7. Fixed a crash that sometimes could happen when using texture mapping.
8. Fixed a crash that could happen when using the steering module with CMS-Flow/CMS-Wave.
9. Made it so you could replace the generic model definition being used without deleting the geometry.
10. Fixed a problem that caused element labels to print very small at times.
11. Fixed an issue where SMS would set the ADCIRC output files to binary if the output information wasn't complete. This made it impossible to get back to ASCII files without editing the files by hand.

SMS 11.0.2

1. Problem selecting cells in the spectral energy dialog.
2. Fixed a crash that could happen if you didn't have a scatter dataset and then converted it to mesh and then to map.
3. Fixed an issue where mesh elements were not displayed according to the setting in the display options dialog after generating a mesh using LTEA.
4. Fixed an issue where SMS stored the wrong directory if the user had to path to find the LeProvost files.
5. Fixed an issue with the LTEA mesh generation feature where SMS was not redistributing the ocean boundary based upon the user specification.
6. Fixed an issue where LTEA had spurious error messages pop-up during the meshing process.
7. SMS would sometimes write the wrong timestep when saving datasets to a tabular data file (*.txt).
8. AdH nodestring symbols were not displayed correctly after loading the simulation until the user went to the boundary condition dialog. This has been fixed.
9. SMS was not allowing CMS-Wave structure cells to have a negative modification value but this is valid in some cases.
10. Fixed an issue with the CMS-Wave model control parameters dealing with spreadsheet rows disappearing and not resizing correctly.
11. Fixed a crash that happened sometimes after duplicating a CMS-Wave grid.
12. Fixed a problem where the clock for first frame in a filmloop was not being displayed correctly if the starting time was not 0.0.
13. Fixed a problem where 1D *.mif files were not imported correctly into SMS when loading a *.tcf file.
14. Fixed a crash with running STWAVE full-plane after loading a project created with an earlier version of SMS.
15. SMS was not correctly interpreting some of the parameters correctly when importing STWAVE model files.
16. Fixed a hang when copy/pasting values into the time-series editor used with AdH.
17. STWAVE executable fixed to write datasets correctly.
18. Zonal classification could sometimes identify polygons incorrectly.
19. Auto-zmag was not updating when a new mesh was created until a frame or similar command was issued.
20. SMS was not correctly remembering specified raster projection after saving/loading a project.
21. In geographic projections, SMS was labeling x locations "east" that should have been labeled "west."
22. The raster contours sometimes changed after going to the display options dialog even when not changing the contour settings.
23. The 2D Gridframe was sometimes hidden behind raster.
24. Typo fixed in the menu for data calculator.
25. STWAVE boundary conditions dialog not "cutting" a row correctly.
26. Fixed crash when loading a project with 1D elements created in an earlier version of SMS.
27. Made it so you can specify paths to sediment datasets in *.xmdf file.
28. Fixed problem displaying functional surfaces with ati card.
29. Changed how LandXML files read to correctly identify "Northing" then "Easting."
30. SMS was incorrectly identifying elements as duplicates and removing them.
31. Fixed problem with local projection in *.mif/*.mid files.
32. Fixed crash when snapping two feature points.
33. STWAVE files not exported correctly when wind datasets being used.
34. Changed how we handle projections read when importing STWAVE model files so non-state plane coordinates are better handled.
35. Converting a raster to scatter created voids in the data.
36. Fixed saving tidal harmonics settings in ADCIRC model control.
37. You can now select all the points in a raster (before the top values sometimes weren't selecting).

SMS 11.0.1

1. Fixed a bug where reprojecting a TUFLOW grid was not updating all of the z values correctly.
2. Fixed an issue where RMA2 mid-side nodes were not retaining their elevation data after saving/loading.
3. Fixed a couple minor issues with the Map→scatter dialog.
4. Made it so SMS would preserve the order of nodestrings from the AdH *.bc file.
5. Fixed a problem with ADCIRC fort.23 files that were reloading incorrectly.
6. Fixed a crash that could occur when copy/pasting when making irregular culverts in TUFLOW.
7. Fixed a problem where the functional surface legend was obscured by other data rather than being above the other data.

SMS 10.1 Intermediate Release Bugfixes

SMS 10.1.11 - Released January 24, 2011

Fixed a problem where doing film loops with cartesian grids with transient datasets used the same time step over and over rather than cycling through the timesteps.

Fixed crash using Map->Scatter.

Fixed an issue where PTM particles would "jump" when going inactive due to improper treatment of NULL values.

Fixed a crash when using ADCIRC<->CMS-WAVE steering module.

Fixed a problem where the volume reported for selected elements is off.

SMS 10.1.10 - Released October 25, 2010

Fixed a problem with converting cartesian grid data to a scatterset.

Fixed a problem where SMS was not saving the georeferencing information for an image.

Fixed zonal classification so it will work correctly with multiple zones and with dynamic solutions correctly.

Fixed a problem where NULL values were not plotted correctly.

Fixed a crash that sometimes occured when going to the page setup dialog.

Fixed a crash that could occur when displaying vectors along nodes in a coverage.

Fixed an issue where the palette used for contour values is not saved and restored with the project.

SMS 10.1.9 - Released September 10, 2010

Fixed an issue with scatter datasets created incorrectly sometimes with data calculator.

Fixed a crash that occurred at times when saving a project after duplicating a scatter set.

Fixed a crash when trying to define a domain using selected vertices.

Fixed an issue with hot starting CMS steering runs.

Fixed a problem where created CGWAVE nodestrings don't get saved with the project.

SMS 10.1.8 - Released July 30, 2010

Fixed a problem where all Bouss2D probes had the results from the first probe.

Fixed a crash interpolating to an ADCIRC grid.

Fixed a problem deleting scatter points.

Fixed a problem where multiple grids could have the same GUID causing one not to get displayed.

Fixed a problem when using Bouss2D and spectra dws files the direction of the wavemaker was not working correctly.

Fixed a problem where SMS would not startup if they moved their "My documents" folder.

SMS 10.1.7 - Released June 17, 2010

Changed the function that copies an STWAVE or CMS-Wave simulation to create a separate copy of the spectra for the new simulation.

Added output files to the data distributed with the RMA2 tutorial

Made a change to where SMS save the log file for TUFLOW to allow non-administrative users to run the model.

Fixed the interactive tool for dragging column boundaries in a CMS grid.

A new update for CMS-Flow resolved a few model crashes that had been reported.

Fixed a bug in mesh generation that occurred when degenerate loops (made out of degenerate arcs) exist in the conceptual model.

Fixed a problem with filtering large dataset using the angle filter.

Fixed a problem related to duplicate node tolerance that occurred when triangulating geographic data.

Fixed the infra gravity wave toolbox for generating input to CGWAVE. This fix also required the distribution of additional utilities in the BOUSS2D folder in the models folder.

Fixed a problem in saving the contour options for a dataset. These were not being preserved from one session to the next with a project.

Fixed a problem interpolating from one scatter set to another.

Fixed a crash that occurred if a user hit the ESC key while saving a scatter set.

Fixed a problem in the color of a grid cell changing.

Fixed a crash in zonal classification.

Fixed a problem when reading ".3dm" files so that the datasets associated with these files are opened automatically.

Fixed a crash that occurred when trying to generate a full circle domain around a single point on a coastline arc.

Fixed the text that appears when you hover over the Select Grid Frame tool.

SMS 10.1.6 - Released May 3, 2010

When saving/loading a FESWMS model the nodestrings do not get duplicated and retain the original direction.

Fixed a problem where some of the FESWMS material properties were not being saved correctly.

Fixed a crash reading some older annotation files.

Potentially Breaking Change - We changed SMS so we no longer provide path information to RMA4 for RMA2 solution files. The issue is that RMA4 does not support very long file names and the binary geometry file has to be in the same folder. We recommend saving/running your RMA4 model in the same directory as you run RMA2. Otherwise, you will need to copy the appropriate files into the same folder as RMA4.

Fixed an issue where a reference to PTM solution files were not being referenced in the project file.

Fixed an issue with appending mesh files that lead to bad values at element centroids.

SMS 10.1.5 - Released March 11, 2010

Fixed the issue with merging meshes that have duplicate or near duplicate nodes deleting border elements and preventing a clean merge.

Fixed a problem where SMS was not saving the PTM setting for "Source and trap z-value relative to datum"

Fixed a problem with the generic model interface that resulted in duplicate global parameter tabs when a 2dm file with the same parameter group was read in.

Fixed an issue that occurred when certain projections were used with Cartesian grids. The problem happened when using the tutorial files with CMS-Flow. The tutorial files have also been changed to the correct projection.

Fixed an issue with display vectors on a grid from CMS-Flow solutions.

Fixed an issue where certain CMS-Flow/CMS-Wave steering files would crash during the steering process if some of the cells along the CMS-Flow boundary condition went dry.

In CMS-Flow the grid is automatically oriented so the origin is in the lower left. This was not working properly and has been fixed.

When adding points to a scatter set, all but the active dataset are deleted. At times the wrong dataset was left. This has been fixed.

SMS 10.1.4 - Released Feb 12, 2010

Fixed a meshing bug that appeared in the last update.

Fixed a problem reading some ADCIRC fort.15 files.

Fixed displaying scatter scalar values colored by contour scheme.

Fixed a BOUSS2D model crash that seemed to happen more with Vista and Windows 7.

SMS now opens project files correctly when you double click on them.

Fixed the data calculator Max(:all) and Min(:all) functions used to determine the max or min of a dataset through all timesteps.

Fixed an issue where the materials boundary lines were not updated correctly when changes were made to the assigned materials.

Fixed an error with the generic model interface reading and writing model parameters.

Fixed some problems with the "Viz" package where features were not accessible like they should be.

Fixed an issue with reading some RMA2 solution files. The files this affected was quite rare.

Fixed a problem with the scatter vertex tolerance in geographic coordinates. This duplicate node tolerance is now always m in geographic coordinates. The conversion isn't precise so it is an approximate value.

Fixed an issue where the label on the initial water surface elevation in a RMA2 simulation was not being updated when the coordinate system of the project changed.

Fixed the graphic and manual for FESWMS wind direction graphics and explanation.

Fixed a problem in the last update where reprojecting cartesian grids removed variable cell widths in CMS-Flow.

SMS 10.1.3 - Released Jan 11, 2010

Fixed a problem where the incident wave condition was not being saved for CGWAVE

SMS defaults ADCIRC fort.73 files to more appropriate names when read in

Fixed a background image shift that was happening in transient animations

Fixed a problem that lead to a bad contour value with CMS-Flow solutions

Reintroduced ability to choose Map to 2D Mesh on an area property coverage to modify the materials in your mesh

Fixed a problem where build polygons was reseting the material types for the polygons

Fixed STWAVE so it ouputs ASCII files when they are selected for output

Fixed a problem reading PTM source files

Fixed a problem where doing a save as project with a PTM simulation referenced the old PTM files

Removed distinguishment of land and water cells in STWAVE and CMS-Wave since no such specification exists in the models. This was causing some cells with solution data to not be displayed.

Fixed an issue with CGWave 1d depth extraction not working.

Fixed a problem where vectors were not saved to a jpeg correctly if the scale was greater than 1.0

Fixed a problem where SMS would sometimes freeze when trying to delete duplicate nodes. This happened particularly when the tolerance was large compared to the average spacing between nodes.

Fixed a problem where CMS-Wave spectra were incorrectly read in as full plane spectra.

SMS now correctly stores the reflection coefficient for CGWAVE

Fixed a problem with displaying cellstrings that had portions with negative z elevations.

SMS 10.1.2 - Released Nov 16, 2009

SMS now reads in fort.61 files correctly.

SMS now creates FESWMS weir IDs correctly.

Fixed a problem where contoured functional surfaces were not displayed correctly.

The feature stamping dialog no longer truncates the words in dialog.

Fixed the roughness by depth image in the FESWMS material property dialogs.

Fixed a problem where you couldn't turn off the legend for contoured functional surfaces.

Readded the capability to do Map->2D Mesh from an area property coverage to copy roughness parameters to an existing mesh.

SMS now correctly updates the screen when triangle edges are swapped without requiring a manual refresh command.

Fixed a problem that profile plots with multiple arcs were not being displayed correctly.

SMS can again show images in the background of flow trace animations.

Fixed a problem with exporting meshes as tabular data files in the 64-bit version.

Fixed a problem where adding scatter points into an existing scatter set caused SMS to hang.

Fixed a problem where converting a scatter contour to an arc broke contour display in other modules.

Fixed some issues with vectors not updating correctly when changing the view.

Fixed an issue where time-series plots of flux were not being computed correctly in the Cartesian Grid Module.

Fixed a problem where splitting a BOUSS2D cellstring caused SMS to hang.

Fixed a problem interpolating vector datasets to a STWAVE grid.

SMS 10.1.1 - Released Oct 26, 2009

Tutorials

New Tutorial – Working with TINs (scattersets) in SMS. Includes guidance on using breaklines and other new features in SMS 10.1

Updated Tutorials – Google Earth tutorial has been updated to show the difference between exporting Raster and Vector KMZ files and to demonstrate how to build KMZ filmloops. The ADCIRC tutorial files have also been updated.

Executables and dlls

The CGWAVE executable for the original installation was incorrect. The correct executable is now being distributed

The correct BOUSS2D executable is now being distributed.

Some of the dll's in the original installation were the incorrect versions. These have been fixed.

Bugs

Windows 7 File Management – There have been some issues with the settings files used by SMS due to inablity to write in the program files area on Windows 7 and/or Vista. These files have been moved to a user area that can be written.

Importing Observation points – Some problems importing observation points have been fixed.

Fixed the “Zoom to “ menu items when right clicking on an object in the project explorer.

Fixed problems related to reproject scattersets.

Fixed a problem with how SMS writes TUFLOW input files when using weirs over structures.

Fixed some problems relating to merging/splitting rows or columns in CMS-Flow.

Fixed functional surface contour legends.

Fixed a problem reading some generic model interface files.

Known Issues

We are currently working to find solutions to the following known issues:

- SMS 10.1 may not run on virtual machines

SMS 10.0 Intermediate Release Bugfixes

Intermediate release SMS 10.0.11 - February 10, 2009

- Fixed a merge meshes crash
- Fixed a problem with RMA2 Marsh Porosity where all of the data was not being saved
- Fixed a problem with ADCIRC weirs
- Fixed a problem with zonal classification
- Fixed a crash in converting GIS shapefiles to feature objects
- Fixed a problem where RMA2 steering it always reports 0.0% of target

Intermediate release SMS 10.0.10 - December 10, 2008

- Fixed a crash with converting Shapefiles (GIS) to Feature objects (Map module)
- Fixed a problem where Shapefile->Map created additional arcs
- Material Assignments to elements mixed up
- Fixed a problem with initial values not initializing correctly in the vector to scalar dialog
- Fixed a crash when opening sms or fpr file
- Fixed a problem where renaming an arc incorrectly closed a profile plot
- Fixed a problem where image pyramids were sometimes not setup correctly for large images
- Fixed a problem where SMS sometimes hung when doing a delete all command
- Fixed a problem where ADCIRC spatial attributes were not being seeded correctly from the specified dataset
- Fixed a problem where ADCIRC spatial attributes were not using the default values which caused larger files to be written then were necessary
- Fixed a problem where reading a partially complete ADCIRC solution was completely aborted rather than reading valid timesteps

Intermediate release SMS 10.0.9 - November 4, 2008

- Fixed a crash with the material properties for generic model interface
- Fixed a problem with contour labels showing incorrect values
- Fixed a problem with the controls to adjust the size of the contour legend
- Fixed a crash with cleaning really complicated map coverages
- Fixed a crash with opening project files with plots that have incomplete data
- Fixed a crash with zonal classification feature
- Fixed a crash that occurred when moving SMS between monitors when using multiple displays
- Fixed a crash with the FESWMS global bed control
- Fixed a problem with demo mode that prevented the user from panning or zooming correctly
- Fixed a problem with using the virtual earth locator to download teraserver images

Intermediate release SMS 10.0.8 - October 6, 2008

- Fixed a problem with merging cartesian grid rows and columns
- Fixed a problem where generating cellstrings along boundary produced overlapping strings

Intermediate release SMS 10.0.7 - August 29, 2008

- SMS now correctly writes the viscosity data to the appropriate input files
- Fixed a problem with RMA4 nodestrings with midside nodes
- Changed find by node ID so that the red circle stays around longer
- Fixed a crash relating to flux plots
- Fixed a crash that occured when reading certain project files
- Fixed a crash relating to boundary conditions with the Generic 2D Mesh Model
- Fixed a crash that occured when running FESWMS
- Fixed a problem with an improper wave angle written out when steering with CMS-Wave
- Fixed a problem with specify input paths for PTM hydrodynamics
- Fixed a crash when selecting a cell in a grid with no elevation dataset
- Fixed a problem where the BOUSS2D Wave generator arrow was displayed in the wrong direction
- Fixed a crash that occured when reading an ADCIRC fort.61 file
- Fixed a crash that occured when observation coverages were deleted before their associated plots

- Fixed a problem with converting Cartesian Grid coordinates
- Fixed so the selection highlighting is done correctly when using the Cartesian Grid Find Cell command.
- Fixed a crash after deleting materials from the CGWave materials dialog
- Updated the PTM executable so it will work with hardware locks
- Fixed a crash when using nested STWAVE grids
- Fixed a crash that occured when you tried to read in scattered data when the scatter module is not enabled
- Fixed a crash when reading incomplete FESWMS .sed files
- Changed SMS to not report PTM warnings for "Mapping Output Interval" if mapping output is turned off
- You will no longer be prompted to save twice when working with STWAVE
- SMS now treats ADCIRC fort.73 and fort.74 files correctly in the model control tab
- Fixed a crash with some RMA4 project files
- Fixed problems with exporting multiple timesteps into a tabular data file
- Fixed a problem with functional surfaces not displaying correctly after changing the display options.
- Fixed an endless loop when trying to create "multiple view" filmloops

Intermediate release SMS 10.0.6 - July 15, 2008

- A problem with the display getting fouled up after changing the z-mag until the next frame command has been fixed
- Fixed crash when trying to assign ADCIRC weirs to nodestrings with different numbers of nodes
- SMS now correctly uses external files specified in the TUFLOW grid options dialog
- Fixed a problem where invalid material references could be created when using the generic model interface
- Fixed a crash that happened on certain computers when saving scatter sets as TINs
- Fixed a problem with the tutorial files for the generic model interface
- Fixed a crash with multiple view filmloops
- Inverting cell selection with Cartesian Grids now works correctly
- Select Cartesian Grid cells by data value has been fixed
- The node ordering for boundaries and forcing files has been made consistent
- STWAVE now runs multiple events from SMS
- The display will now update correctly after changing a nodes x, y coordinates
- Fixed a crash with observation fluxes
- Fixed a problem with observation attribute names for arcs when using both nodes and arcs
- Scatter edges now display correctly when panning after swapping edges
- The display now updates correctly after reversing the direction of a feature arc
- Fixed a crash with zonal classification if you enter the dialog with zones defined that don’t have any criteria
- Fixed Ctrl+ key shortcuts to match the descriptions in the menu
- When mapping an elevation dataset the correct name is now used
- Fixed a problem saving TUFLOW event data
- Select by material type now refreshes appropriately
- Fixed select by dataset value for nodes
- Fixed a crash with certain FESWMS sed files
- Fixed a crash that occurred sometimes when loading a DEM from an asc file
- Fixed a crash that occurred with Cartesian Grid functional surfaces and ATI graphics cards
- Modified the Scatter Contour -> Feature dialog to work more cleanly
- Fixed a crash that occurred with reading some ADCIRC datasets
- Fixed creating cell strings for PTM from CMS-Flow

Intermediate release SMS 10.0.5 - June 20, 2008

- Fixed a problem with SMS accessing deleted temporary files.
- Fixed a crash with trying to access plot data after deleting feature arcs.
- Bouss-2D wave maker images now appear right side up.
- Fixed a crash with deleting nodes with nodal boundary conditions being displayed.
- Fixed a problem with creating PTM input data from an RMA2 solution.
- Fixed a crash that sometimes occured when smoothing size datasets in the mesh module.
- Fixed a problem with reading RMA2 mesh and solution files that have gaps in the node ids.
- Fixed a crash that occured when deleting improperly created Cartesian grid cell strings.
- Fixed a crash with exporting tabular data.
- We now give an explanation message if we cannot create a nautical grid.
- Fixed a problem where sometimes multiple scalar and vector datasets could appear active.
- SMS now updates the time window when datasets are referenced to a specific timeframe.
- Fixed a problem where SMS doesn't terminate on exit if a solution dataset has no valid values.
- Coordinate transforms are now applied to all the grids and not just the active grid.
- Fixed a meshing crash that sometimes happened with scalar density paving.
- SMS now writes the TUFLOW initial water level value in the tgc file so it can be overriden by water level polygons.
- SMS now writes the STWAVE nesting cells correctly.
- Fixed a problem where it was necessary to launch STWAVE twice before it would run successfully.
- Fixed a problem with Bouss2D probe dialog.
- SMS now saves the Cartesian Grid arc options.
- Changed the output format of the CMS Wave energy file to ensure proper spacing between numbers.
- The Refine point dialog now uses I and J rather than U and V to be consistent with other parts of the interface.
- SMS now handles the TUFLOW external grid file correctly.
- Trying to assign weirs to nodestrings of different numbers of nodes no longer crashes.
- Fixed a problem with changing the z-magnification that broke the display until the next frame occured.

Intermediate release SMS 10.0.4 - June 11, 2008

- Mesh contours will now display correctly when there are timesteps that are completely dry
- Fixed a CMS-Flow, CMS-Wave steering crash
- Fixed a crash that occured when creating arrowheads using the map draw tools
- Fixed GIS to Coverage when using a TUFLOW network coverage
- The profile plot now updates when adding and removing arcs from arc groups
- Map nodes were drawn in their old location after being dragged to a new location
- Arcs would sometimes disappear behind transparent contours
- The origin of observation arcs now save with map files
- Default stamp elevation for feature stamping is now used correctly
- Newly created coverages are now active in the project explorer
- Transparency for Cartesian Grid functional surfaces has been fixed
- Color filled and linear contours now appear at the same z elevation for Cartesian Grids
- Naming observation arcs no longer cause a crash
- Fixed a nodestring display update issue
- Fixed a crash reading shape files
- PTM post processing datasets are now saved with the project
- Triple clicking the same point when creating an arc no longer causes a crash

- Having multiple CMS Flow grids with cellstrings no longer causes bad cellstrings
- Trying to save a scatter set to a read only file no longer crashes SMS
- SMS again allows users to drag files onto the icon to open them
- An issue with the timestep window not showing up when using PTM has been fixed
- Fixed a crash when deleting points used by an observation plot
- There were times when newly created scatter points were not added to the triangulation. This has been fixed.
- At times SMS would hang when optimizing the triangulation
- Contour labels are no longer cut off by color filled contours
- Fixed a selection problem with scatter vertices by when using a polygon
- Some DEM files were not reading in correctly but they do now
- TUFLOW 1D Flow vs WSE boundary conditions were not be written to the TUFLOW files correctly
- The get info dialog for meshes no longer reports information from unused models
- Fixed the DEM import dialog so that the feedback is correct and makes sense
- Fixed a problem with display updates when assigning flowrate boundary conditions
- Functional surfaces now frame correctly
- The screen refreshes correctly when interpolating between nodes
- It is now always possible to exit selecting with a box by using the escape key
- Fixed a crash that occurred when copy a coverage with selected entities
- Removed excess information from the zonal classification logs
- Fixed a crash when reading an invalid RMA2 solution
- SMS now stores Bouss2D grids the right size and with the correct indices
- Fixed some problems with Bouss2D probes
- SMS now displays coverages in the correct order
- Fixed a problem where multiple CGrids sometimes displayed
- Fixed 1D element contours
- Fixed problems with grid frames (dragging, deleting, and copying)
- Fixed a problem with 1D element node updates
- Coastline files can be created even if you don't have CGWave enabled
- Fixed a crash with deleting cell strings

Intermediate release SMS 10.0.3 - May 1, 2008

- PTM executable updated to version 2.025
- Saving out coastline file available if CGWAVE or ADCIRC is enabled
- ADCIRC 63/64 file crash
- Slow deletion of feature points
- STWAVE coverage issue specifying land polygons
- GIS to Feature data crash
- Generic model interface - global parameter definition bug
- Import wizard problem where z elevations not properly mapped to feature points
- Fixed multiple views film-loop
- GIS to Map for TUFLOW network coverage fixed
- Relocated map node displayed at old location until refresh
- Origin of observation arc is now saved in the map file
- Cartesian grid Color fill and linear contours are now displayed at consistent elevations to improve visualizing them in 3D
- Name observation arc crash
- Nodestring display issue

- Fixed create arc tool crash
- Editing scatter points and overwriting HDF5 file problem
- Crash when deleting feature points in specific situations

Intermediate release SMS 10.0.2 - March 14, 2008

- Crash when performing change/delete when triangles selected
- Print preview not functioning
- Scale legend does print correctly
- Text and drawing objects not printing correctly
- Transform feature objects was not working for PTM coverages
- STWAVE Boundary conditions types not saved correctly

Intermediate release SMS 10.0.1 - February 19, 2008

- Fixed the inability to select Southern hemisphere UTM zones
- Fixed several graphical glitches including texture mapping on a surface and several update problems
- Fixed various problems with the spectral energy dialog
- Removed misleading messages that originate from the generic model interface
- Fixed a crash that sometimes occurred when using zonal classification
- Fixed a problem with PTM traps that was reporting the open/closed state incorrectly
- Fixed some issues with models that only support metric units
- Fixed a crash that sometimes occurred when trying to use LTEA
- Changed STWAVE to write out all datasets by default
- Fixed coordinate conversions for PTM sources
- Fixed a bug that sometimes caused poor meshes to be generated

Related Topics

- What's New in SMS
- Downloads
- Installing and Setting up SMS
- System Requirements
- License Agreement

Editing the view with the Mouse

The user can now navigate easily with mouse controls. The three controls include:

- If your mouse has a middle button (or a mouse wheel), you can scroll the wheel to zoom in and out.
- If your mouse has a middle button (or a mouse wheel), you can hold it down and drag to pan the view.
- You can also hold down both the right and left mouse buttons and drag to rotate the view.

FHWA:2010 Webinars

Federal Highways Administration and Aquaveo have partnered to present a series of webinars on using SMS/WMS. This page contains links to past webinars that you can watch.

January: Introduction to WMS: How to Build a Model using the Hydrologic Modeling Wizard

Slide Presentation [1] (50 minutes)

Software Demonstration [2] (20 minutes - please note that the sound for this recording will take a few seconds to come up so don't worry if you don't hear anything initially)

February: Introduction to SMS models FESWMS and TUFLOW

Watch (45 minutes) [3]

March: Using coordinate systems in SMS and WMS

Watch (50 minutes) [4]

April: Data Collection in SMS

Watch (53 minutes) [5]

May: Data Collection in WMS

Watch (1 hr 8 min) [6]

June: Editing Scattered Data in SMS

Watch (47 min) [7]

July: Using the Hydrologic Modeling Wizard in WMS

Watch (55 min) [8]

August: Flow Modeling using SMS and TUFLOW

Watch (1 hr 27 min) [9]

September: HEC-RAS Conceptual Modeling in WMS

Watch (1 hr) [10]

October: Setting up FESWMS Models in SMS

Watch (1 hr) [11]

November: Mapping Floodplains in WMS

Watch (1 hr) [12]

December: FESWMS(and TUFLOW), Modeling structures in 1D and 2D

Watch [13]

Back to XMS

References

[1] http://fhwa.na3.acrobat.com/p23061878/
[2] http://fhwa.na3.acrobat.com/p11298550/
[3] http://fhwa.na3.acrobat.com/p22046632/
[4] http://fhwa.na3.acrobat.com/p27781810/
[5] http://fhwa.na3.acrobat.com/p94892485/
[6] http://fhwa.na3.acrobat.com/p27287621/
[7] http://fhwa.na3.acrobat.com/p11032223/
[8] http://fhwa.na3.acrobat.com/p63062182/
[9] http://fhwa.na3.acrobat.com/p70798648/
[10] http://fhwa.na3.acrobat.com/p45592209/
[11] http://fhwa.na3.acrobat.com/p11206748/
[12] http://fhwa.na3.acrobat.com/p82094152/
[13] http://fhwa.adobeconnect.com/p62228745/

General Interface Features

This article addresses features that do not belong to specific modules.

Display

- Editing the View with the Mouse
- Display Options
- Visualization

Data that does not belong to a specific module

- CAD Data
- Annotation Layers
- Images
- LandXML Files

Misc

- Coordinates
- File Formats

Keyboard Shortcuts

Many commands in SMS are can be accessed using keyboard shortcuts.

Standard Menus

Keyboard Shortcuts

Modifier	Key	Command
	F	Display | Frame Image
	Space	Refresh | Refresh
Ctrl	C	Edit | Copy to Clipboard
Ctrl	V	Edit | Paste Tabular Data
Ctrl	A	Edit | Select All
Ctrl	O	File | Open
Ctrl	P	File | Print
Ctrl	S	File | Save Project
	F1	Help Menu | SMS Help
	F2	Pan
	F3	Zoom
	F4	Rotate
Shift	F	Display | View | Front
Shift	O	Display | View | Oblique
Shift	V	Display | View | View Options
Shift	P	Display | View | Plan
Shift	Z	Display | View | Previous
Shift	S	Display | View | Side

Related Topics

- Right Click Menus
- Layout of the Graphical Interface

Publications

SMS Related Publications and Reports

The following is a partial list of publications and reports related to the use of SMS. Please feel free to make additions that have not been listed.

WES reports are available through the Interlibrary Loan Service from the US Army Engineer Waterways Experiment Station (WES) Library, telephone number (601) 634-2355. National Technical Information Service (NTIS) report numbers may be requested from WES Librarians. To purchase a copy of a report, call NTIS at (703) 487-4780.

External Lists of Articles

- ADCIRC publications [1]
- Journal articles using SMS by Prof. Greg Pasternack, UC Davis [2]

2008

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [4]

2007

- SRH-2D Training Presentation [5]
- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [6]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [7]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [8]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User’s Guide [9]
- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User’s Guide [9]
- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [10]
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington" [11]

2006

- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [12]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model [13]
- Aug 2006 ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [14]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [15]
- FISC 2006 Short Course Presentation [16]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [17]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [18]
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project." [19]

- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006)."Analysis of sediment transport following removal of the Sandy River Delta Dam." [20]
- Bountry J.A. and Lai, Y.G. (2006)."Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." [21]

2005

- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [22]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [23]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [24]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [25]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [26]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerodible (Hard) Bottom in Two-Dimensional Morphology Change Models [27]
- 2005 US-China Workshop Paper [28][29]

2004

- May 2004 ERDC/CHL TR-04-2 Two-Dimensional Depth-Averaged Circulation Model M2D: Version 2.0, Report 1, Technical Documentation and User’s Guide [30]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [31]

2003

- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [32]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations[33]

2002

- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [34]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [35]
- Mar 2002 ERDC/CHL CHETN-IV-40 Guidelines for Using Eastcoast 2001 Database of Tidal Constituents within Western North Atlantic Ocean, Gulf of Mexico and Caribbean Sea [36]
- Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [37]

2001

- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [34]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [38]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [39]
- Mar 2001 Technical Report CHL-98-32 Shinnecock Inlet, New York, site Investigation Report 4, Evaluation of Flood and Ebb shoal Sediment Source Alternatives for the West of Shinnecock Interim Project, New York [40]

1999

- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [41] [42]

1998

- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [43]

1990

- Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [44]

References

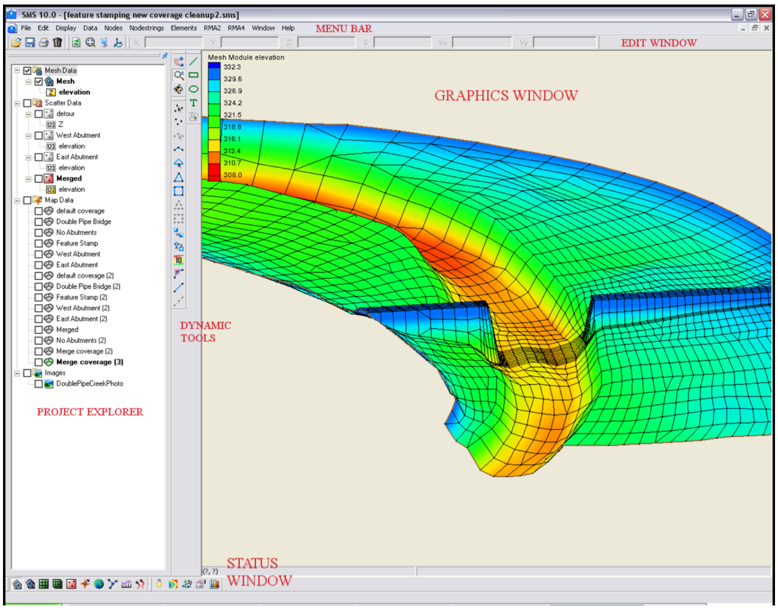
[1] http://adcirc.org/Related_publications.html
[2] http://shira.lawr.ucdavis.edu/publications.htm
[3] http://www.fsbpa.com/08Proceedings/07AlymovTruittPoffAnderson2008.pdf
[4] http://cirp.wes.army.mil/pubs/chetns/CHETN-IV-71.pdf
[5] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/SRH2D-v2-Training-Dec2007-Taiwan.pdf
[6] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-76.pdf
[7] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-75.pdf
[8] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-69.pdf
[9] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-73.pdf
[10] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-74.pdf
[11] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Dungeness%202dmodel%20report%20May7_07.pdf
[12] http://www.waveworkshop.org/9thWaves/Papers/Smith.pdf
[13] http://www.dtic.mil/cgi-bin/GetTRDoc?AD=ADA455437
[14] http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf
[15] http://cirp.usace.army.mil/pubs/chetns/CHETN-III-73.pdf
[16] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2006-April-GSTARW-Training.pdf
[17] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-71.pdf
[18] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-67.pdf
[19] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report-Elwha%20Nov%2014%202006.pdf
[20] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Sandy%20River%20Report_031706.pdf
[21] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Final%20Report%20Savage%20Rapids%20Jan%2023%202006.pdf
[22] http://el.erdc.usace.army.mil/elpubs/pdf/doerd4.pdf
[23] http://el.erdc.usace.army.mil/elpubs/pdf/doerd5.pdf
[24] http://el.erdc.usace.army.mil/elpubs/pdf/doerd6.pdf
[25] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-70.pdf
[26] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-69.pdf
[27] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-63.pdf
[28] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2005-US-China-Workshop-paper.pdf
[29] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Paper%20US-China-Workshop%202005.pdf
[30] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS!155
[31] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-68.pdf
[32] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-60.pdf
[33] http://chl.erdc.usace.army.mil/Media/4/9/5/chetn-i-67.pdf
[34] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-i-66.pdf
[35] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-41.pdf
[36] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-40.pdf
[37] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-ii-45.pdf
[38] http://chl.erdc.usace.army.mil/Media/5/6/9/BOUSS-2D.pdf
[39] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-32.pdf
[40] http://www.adcirc.org/publications/2001/2001_Militello03.pdf
[41] http://www.adcirc.org/publications/1999/1999_Militello.pdf

[42] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/cetn-iv-21.pdf
[43] http://champs.cecs.ucf.edu/Library/Research_Reports/pdfs/
CGWAVE,%20A%20coastal%20surface%20water%20wave%20model%20of%20the%20mild%20slope%20equation.pdf
[44] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/cetn-ii-21.pdf

1.2. Layout

Layout

At a glance



SMS Layout

- The project explorer shows data currently loaded in project
- Menu bar depends upon the active module and model
- Edit window show x, y, z, scalar, and vector values
- Edit window values can be edited in some circumstances
- The status window on the bottom of the graphics window shows coordinates and selection information
- Help information is displayed at the bottom of the SMS screen
- Several toolbars are used in SMS. The dynamic tools change based upon the current module.

The interface to SMS has been designed in a modular fashion. Separate modules are used for each data type. As the user switches from one module to another, the available menus and tools change. Inside the modules, the user associates a numeric model with a mesh or grid. When that grid is active, the tools and menus for the associated model are also enabled.

The SMS screen includes several toolbars, edit fields, and menus. Some of these change as the user switches modules or numerical models. The principal components include:

- Menu Bar – Menu commands to issue commands. These change as the module and model change.
- Edit Window – Fields directly below the menu bar showing the coordinates and function values for selected entities.
- Graphics Window – Display panel to show the data being manipulated.
- Project Explorer (Data Tree) – Tree representation of all the data currently referenced through SMS.
- Time Step Window – Appears when transient data is available.
- Toolbars – Several toolbars can be displayed. For more information on each toolbar, see the Toolbars article.
- Help or Status Window

The toolbars, project explorer, time steps window and edit window are dockable windows. Dockable windows may be positioned by the user.

Data Toolbar

The Data Toolbar contain tools to query or obtain data. Whether the Data Toolbar appears at startup is set in the Preferences Dialog. The following tools are available in the Data Toolbar:

Tool	Tool Name	Description	Right Click Menu
	Measure Tool	The Measure Tool is used to measure distances interactively. The units used to report the measured distance is specified on the Toolbars tab of the Preferences Dialog.	N/A
	Get Data Tool	The Get Data Tool is used to specify the location of interest for obtaining data using the import from web feature.	N/A

Related Topics

- Layout of the Graphical Interface

Dynamic Tools

The Dynamic Toolbar contains tools that apply to the selected module and active numerical model. These tools are called dynamic because the available tools change whenever the module or numerical model is changed. These tools are used for creating and editing entities specific to the module. They appear between the Project Explorer and the Graphics Window below the Static Tools.

Selection Tools

The selection tools in SMS allow users to select entities displayed in the *Main Graphics Window*. It is necessary to first select objects before issuing many of the commands in SMS. For example, to delete a node, the node must be be selected and then the *Delete* command issued. Selections can be made using a box, polygon, arrow, or by clicking a single location. In addition, selections can be toggled, have new items added, or remove items. Below is a list of modifier keys and corresponding actions.

- **None** – This will clear the current selection and add the newly selected items. Dragging will create a selection box. All items contained in the box will be selected.
- **Ctrl** – Clicking while holding the *Ctrl* key will create a polygon. All items contained in the polygon will be selected. If control is held while dragging, an arrow will be created. All items which the arrow passes through will be selected. (*Control* will cause the same behavior with any combination of the *Alt* and *Shift* keys)
- **Shift** – Holding *Shift* causes all newly selected items to be toggled. If it was selected before it will be unselected, and if it was not selected it will be selected.
- **Alt** – Holding the *Alt* key causes all newly selected items to be added to the selection list regardless of previous state.
- **Alt + Shift** – Holding *Alt* and *Shift* causes all newly selected items to be removed from the selection list regardless of previous state.

The various selection types, polygon, arrow, and box, are available in all tools with the exception of the arrow. An arrow selection can only be performed when selecting line or polygon (e.g. mesh elements, scatter triangles, etc) elements. The arrow must cross a polygon or line edge to select it.

When selecting polygon features the rules for selection may vary slightly. In the map module all vertices of the polygon must be contained in the selection box or polygon. For mesh elements, scatter triangles, and Cartesian grid cells only the centroid must be contained.

When clicking a single location, the element closest to the eye (i.e. drawn on top of other elements) will always be selected. All other forms of selection (box, polygon, and arrow) will select all elements meeting the required criteria.

Other commands for selecting multiple objects such as *Select With Poly*, *Select by Material Type*, and *Select by Data Value* can be found in the Edit menu.

Related Topics

- 2D Mesh Module Tools
- Cartesian Grid Module Tools
- Scatter Module Tools
- Map Module Tools
- GIS Module Tools
- 1D River Module Tools
- 1D Grid Module Tools
- Particle Module Tools

Edit Window

The Edit Window lies above the Graphics Window and below the Menu Bar. It includes a rows of edit fields and text strings. The edit fields are dim and the text strings blank if nothing is selected. When an entity, such as a mesh node, is selected, the controls display the attribute values of the selected entity. Some attribute values can be edited as shown in the table below. The attribute values are changed by typing in new values and hitting the *ENTER* or *TAB* key. If more than one entity is selected, only the Z edit field is available for editing. Entering a new value in the Z edit field will modify the bathymetry or depth of each of the selected entities. This allows the user to quickly model a feature such as a dredged channel or embankment.

Entity	X Edit Field	Y Edit Field	Z Edit Field	S Edit Field	Vx Edit Field	Vy Edit Field
Mesh Node	Editable for single selection	Editable for single selection	Editable	Editable if an ADCIRC Spatial Attributes data set	Not editable	Not editable
Mesh Nodestring	N/A	N/A	Editable	N/A	N/A	N/A
Cartesian Grid Cell	Not editable	Not editable	Editable	Editable if an CMS-Flow hard bottom or roughness data set	Not editable	Not editable
Cartesian Grid Cellstring	N/A	N/A	N/A	N/A	N/A	N/A
Scatter Point	Not editable	Not editable	Editable	Not editable	Not editable	Not editable
Feature Point	Editable for single selection	Editable for single selection	Editable	N/A	N/A	N/A
Feature Vertex	Editable for single selection	Editable for single selection	Editable	N/A	N/A	N/A
Feature Arc	N/A	N/A	Editable	N/A	N/A	N/A

Related Topics

- Layout of the Graphical Interface

Graphics Window

The *Main Graphics Window* is the biggest part of the SMS screen. The *Graphics Window* is where SMS displays two and three-dimensional data. It is also where the user interacts with that data in SMS. The selected tool in the determines the type of interaction that can be performed in the *Graphics Window*. For example, if the *Create Node* tool is currently selected any click in the *Graphics Window* will result in the creation of a node at the location of the click.

The user has control of what data appears in the *Graphics Window*, and how each data type is formatted. Each type of entity has an associated set of display attributes. These attributes include visibility, color, line thickness, and font type. Each data type is associated with a specific module and the attributes for that type are controlled via that modules *Display Options* dialog.

The *Graphics Window* is integral in the creation, editing and visualization of two-dimensional finite element meshes and two-dimensional finite difference grids. It is also the main means of interacting with a conceptual model and site maps.

The row at the bottom of the graphics window tracks the coordinates and functional values of the location of the cursor. The z coordinate corresponds to an interpolated elevation value from either the mesh or grid, depending on which module is active.

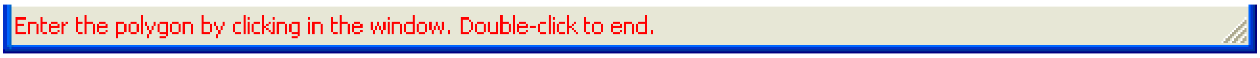
Related Topics

- Layout of the Graphical Interface

Help or Status Window

There are two status bars: one at the bottom of the SMS application window and a second attached to the *Main Graphics Window*. The status bar attached to the main application window shows help messages when the mouse hovers over a tool or an item in a dialog box. At times, it also may display a message in red text to prompt for specific actions, such as that shown in the figure below.

The second status bar, attached to the *Main Graphics Window*, is split into two separate panes. The left shows the mouse coordinates when the model is in plan view. The right pane shows information for selected entities.



Status messages appear in bold red text here to inform the user on progress of a command.

Related Topics

- Layout of the Graphical Interface

Macros

The Macro Toolbars contain buttons to perform frequently used menu commands. All macros are shortcuts for menu commands. Which macro toolbars appear at startup is set in the Preferences Dialog. The macro toolbars include:

Optional Macro Toolbar

- **Lighting Options** – See Lighting Options.
- **Contour Options** – See Contour Options.
- **Vector Options** – See Vector Options.
- **Get Module Info** – See Get Info.
- **Plot Wizard** – See Plot Wizard.

File Toolbar

- **Open** – See Open.
- **Save Project** – See Save Project.
- **Print** – See Print.
- **Delete** – Delete the selected items. If none are selected, delete all items.

Display Toolbar

- **Refresh –** See Refresh.
- **Frame –** See Frame.
- **Display Options –** See Display Options.
- **Plan View –** See Plan View.

Related Topics

- Layout of the Graphical Interface

Menu Bar

Many commands in SMS are accessed through pull down menus located in the menu bar. Each menu can be accessed either with the mouse or by pressing the *ALT* key and the corresponding letter underlined in the menu title. Once a menu is visible the individual commands can be selected with the mouse or by again pressing the corresponding letter underlined in the menu command.

The menus available at any time are dependent on the active module and current numerical model. The first three menus, File, Edit, and Display, are always available. The remaining menus change with the module and the model. This is to partition the available commands into usable groups and avoid unnecessary complexity.

Standard Menus

- File Menu
- Edit Menu
- Display Menu
- Web Menu
- Window Menu
- Help Menu

Module Specific Menus

- 2D Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module

Model Specific Menus

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic Model
- GenCade
- PTM
- STWAVE
- TABS
  - RMA2
  - RMA4
- TUFLOW

Related Topics

- Right Click Menus
- Layout of the Graphical Interface
- Keyboard Shortcuts

Project Explorer

The Project Explorer (which could also be referred to as a "Data Tree") is a dockable window that appears by default on the left side of the SMS screen. This window displays a hierarchical tree structure representing all of the data currently being managed in an SMS simulation. The project explorer includes the following functionality:

Data Representation

The data tree includes one "Module type" folder for each type of data, including:

- Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module
- Images
- CAD Data

Each module type folder in the *Project Explorer* may contain several sub folders. For example, a simulation may includes several scattered datasets, each of which would consist of a folder inside the "Scatter Sets" folder. Further, all data associated with a specific scatter set, such as datasets of elevation or water level, are displayed as entities inside the scatter set folder. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the *Project Explorer*. Folders can be created by right clicking and selecting *New Folder* in the right click menu. A dataset or folder can be deleted simply by selecting the folder and selecting the *Delete* key or by right-clicking on the item and selecting the *Delete* option in the right click menu.

Datasets

The project explorer also includes a list of the datasets associated with each geomtric object (mesh, grid, scatter set). These are displayed below the object in the project explorer and can be arranged into folders.

Module Selection

There are several ways to switch from one module to another. These include:

- Select an entity in the *project explorer*. The module containing the active entity becomes active.
- Right click on the *project explorer* and select the *Switch Module* command.
- Click on the module icon in the module toolbar. The module toolbar is displayed at the bottom of the project explorer by default.

(Note: Switching modules should not be confused with changing the current model inside of a module. When a new model is selected, the tools and menus may change, and the data will be converted as much as is possible. However, some data may be lost.) More Info...

Object Visibility Options

A toggle box appears to the left of each object in the project explorer. This toggle allows the display of all entities associated with the object to be turned on or off. When the toggle is turned on, only items turned on the the object's Display Options are shown.

Right Click Menus

Right click menus are used to interact with data in the *Project Explorer*. See the article Project Explorer Right Click Menus for more information.

Related Topics

- Layout of the Graphical Interface

Project Explorer Right Click Menus

The following Project Explorer right mouse click menus are available based on where the right mouse click is performed.

Project Explorer White Space Right Click Menus

Right clicking in the white space of the Project Explorer invokes an options menu with the following options:

- **Switch Module –** Use to change the active module (active menus and tools are based on the current module).
- **New**
  - **TUFLOW Simulation –** Creates a new TUFLOW simulation.
- **Convert to CAD –** Conversion of visible entities to CAD format. CAD layers are shown in a CAD Data folder in the Project Explorer.
- **Collapse all –** Collapses all items in the Project Explorer.
- **Expand all –** Expands all items in the Project Explorer.
- **Check all –** Checks all items in the Project Explorer. Checked items are displayed.
- **Uncheck all –** Unchecks all items in the Project Explorer. Unchecked items are not displayed.
- **Preferences –** Allows the user to set the Program Preferences in the Preferences Dialog.

Module Right Click Menus

General Options

The following are available for all Module Items:

- **New Folder –** Creates a new folder beneath the module item which can be used to organize datasets.
- **Delete –** Delete the module item.
- **Duplicate –** Duplicate the module item (cartesian grid, scatter set, etc.) including model parameters, boundary conditions, etc.
- **Rename –** Rename the module item.
- **Convert –** Convert the module item to another data type (e.g. Mesh → Scatter Set, Map → 2D Mesh, etc)
- **Reproject –** Reprojects the module item to a different projection.
- **Metadata –** View or modify the metadata associated with a module item.
- **Zoom To [Module Item] –** Reframe the image based on the module item extents.

Module Specific Options

See the module right click menu article for more information on module specific right click menus:

- 2D Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module
- CAD Data

Dataset Right Click Menus

Right clicking on a dataset in the Project Explorer invokes an options menu with the following options:

- **Delete –** Deletes the selected dataset(s). This command may not be available for all datasets. If a dataset has been defined as an input dataset from a model parameter dialog, it must be deleted by changing the model parameter that requires the dataset as an input. It is not recommended that you delete datasets that are part of a single solution file since SMS reads these as a set from the single file.
- **Rename –** Rename the selected dataset. This is an option for datasets stored as part of the SMS project or created in the dataset toolbox. If the dataset comes from a numerical simulation solution, the name will revert to the name specified by the solution when it is read again.
- **Export –** Exports the selected dataset using the Export Dataset Dialog.
- **Scalars to Vector –** Convert two scalar datasets to a single vector dataset. This command only appears on scalar datasets. This operation can also be accessed in the dataset toolbox.
- **Vector to Scalars –** Covert a single vector dataset into two scalar datasets (magnitude and direction or Vx and Vy). This command only appears on vector datasets. This operation can also be accessed in the dataset toolbox.
- **Dataset Contour Options –** Opens the Dataset Contour Options Dialog.
- **Metadata –** Opens the Dataset Metadata Dialog, used to add or view metadata associated with the project. SMS associates the specific data with the selected dataset.
- **Info –** Opens the Dataset Info Dialog which displays characteristics of the dataset. These characteristics include statistics such as maximum, minimum, and range as well as mean and standard deviation.
- **Time Units and Reference –** For a transient dataset, the display of time values in the Time Step Window is controlled by the Time Settings.

Folder Right Click Menus

- **New Folder –** Creates a new folder beneath the selected folder which can be used to organize datasets.
- **Delete –** Deletes the selected folder(s).
- **Rename –** Rename the selected folder.

Related Topics

- Menu Bar

SMS Menus

File Menu

The File menu is one of the standard menus available regardless of the current module and model. The File menu includes the following commands:

Open

The Open command is used to read any file used by SMS. This includes a large selection of file formats, both generic and model specific. This command opens a file browser from which one or more files can be selected. SMS attempts to recognize the file type based on the file extension. The available file formats (extenstions) varies based on the module and model being used. For example, there are several types of *.dat file that are used by different models. If a selected file does not match the anticipated type, a message is given to the user and the user may specify another format type to use to read the file. Data from the file is added to the current data base and SMS updates the display.

Save Project

The Save Project command is used to save an SMS Project File (file extension .sms). The first time this command is invoked, the user is prompted for a file name (unless the user has already opened a project file into SMS). Every other time, SMS saves the project file using either the file name used to save the project or the filename of a project opened in SMS. To save a project with a new name, the Save as... command is used. The SMS Project File is saved as an XMDF file. The contents of the SMS Project File can be viewed using a HDF5 file browser or editor.

Save <Model>

The name of this command changes according to the active module and active model (i.e. Save RMA2 for the Mesh Module, RMA2 model). This command is similar to the Save Project command. The first time this command is invoked, the user is prompted for a file name (unless the user has already opened a model file into SMS). Every other time, SMS saves the model file using either the file name used to save the model or the filename of a model opened in SMS. To save a model with a new name, the Save as... command is used.

Save as...

The Save as... command allows the user to save data currently in the SMS database in a format not associated with the current model nor SMS or to save a model or project file with a new name. The user specifies the Save as type in the Save dialog to specify the file format. The Save as type available at any time depend on the data currently in SMS and on the current module and model (i.e. to save a map file, the user must be in the map module).

Delete All

The Delete All command deletes all the data associated with all modules. It resets the status of the program so that all display option and default values match the values in the "settings" file. This command should be selected when a new modeling problem is started.

View Data File

Since the process of numerical modeling often utilizes many input files and generates many output files, it is not uncommon to review an ASCII data file. When the View Data File command is selected, SMS asks the user to select a file. Based on the Preferences settings, the file will then open or SMS will ask which editor to open the file in. A separate process is created for editing/viewing the selected file using the selected editor. It should be remembered that this is now a separate process and the data in the file is not part of the SMS database. The data may be saved and incorporated into SMS using file read and import capabilities.

 Get Info

The Get Info command reports basic information concerning the data type associated with the active module. For example, for meshes, the Get Info dialog reports the number of nodes, the number of elements, the number of linear elements, etc. For more information, see Information Dialog.

Info Options

When entities are selected, various information about the entities can be displayed or saved. For example, when two nodes are selected, the distance between them can be shown. The values are displayed by default in the Status Window. However, since space along the bottom of the window is limited, as due to the fact that the information being displayed be useful, the user has the option of displaying the information to a separate window and echoing the information into a file. The Info Options dialog allows you to turn on and off various data that can be displayed on the selected entities. The echo of the information to a file can be turned on and off with on Echo to File toggle. When this toggle is turned on, the user is prompted for a file name. The Display Echo Window option opens a window where information is also displayed.

Save Settings

The Save Settings command is used to save the current settings of the program (display options, defaults, etc.) to a default settings file. SMS reads the "default settings" file each time it is launched or the new command is invoked.

Page Setup

Page Setup is used to set printing options. The dialog contains three tabs:

- **Margins**

This page allows you to set up the margins that will be used for printing to the selected printer. The right side of the display shows a gray region representing how the graphics window would be positioned on a printed page. The scale of the printed image directly depends on the margins that are set on this page. The margins have a lower limit depending on the system’s default printer. The Maintain aspect ratio option should usually be checked. When this option is turned on, the size of the printed image be constrained by one pair of margins, either the top and bottom or the left and right.

- **Paper Size**

This page allows you to define the size of the paper for the selected printer. The specific available options are dependent on the system’s default printer. Both the paper size and paper source can be specified, as well as the image orientation on the paper. The preview window shows a sample of what the printed image will look like.

- **Options**

This page allows you to define a scale to be added to the bottom of the printed image. If you change the scale value, the margins are updated to match. The scale is defined as either one inch or one centimeter, and will equal the specified number of units that your data are in. This could be feet, meters, lat/lon, etc. If, after the scale is set, you zoom in or zoom out, the scale will change to match the new world boundaries and the page margins. The preview window shows a sample of what the printed image will look like.

The data displayed in the Graphics Window is then printed through the Print menu item.

Print

The Print command opens the Windows Print dialog. Pages are printed with the data displayed in the Graphics Window using the settings set through the Page Setup menu command.

Demo Mode

Since some users may not require all of the modules or model interfaces provided in SMS, modules and model interfaces can be licensed individually. The icons for the unlicensed modules or the menus for unlicensed model interfaces are dim and cannot be accessed. The Demo Mode command provides a way of evaluating additional modules you may wish to consider licensing in the future. This is particularly useful when using the tutorials provided with SMS. When the Demo Mode command is selected, all modules of the program will be enabled. The only exceptions are that the Print and Save options will be disabled. It is important to note that when the mode is changed all current data will be deleted. When the program is in demo mode, a check mark appears next to the menu item. To return to normal operating mode, select the Demo Mode command again. If an evaluation copy of the software is being used, or if all modules are enabled, this menu item is unavailable.

Register

For more information on the Register command, see the article Registering SMS.

Recent Files

SMS remembers the last five files opened during operation. These files are added to the File Menu. A file can be reopened by choosing it from the list.

Exit

The Exit command is used to exit the program. If your data has not been saved, SMS warns you before it exits.

Edit Menu

The Edit menu is one of the standard menus and is available in all of the modules. The commands in the Edit menu are used to select objects, delete objects, and set basic object and material attributes.

Delete

The Delete command is used to delete the selected objects. This command is equivalent to hitting the Delete or Backspace keys on the keyboard. If no objects are selected when the Delete command is executed, then all of the objects of the tool selection type will be deleted. Unless the Confirm Deletions option is turned on, you will not be asked to confirm the deletion of selected entities.

Select All

The Select All command selects all items associated with the current selection tool.

Select With Poly

The Select With Poly command selects items associated with the current selection tool which are inside a user defined polygon. Create the polygon after selecting the command by clicking in the Graphics Window. The polygon is closed with a double-click. A similar feature called "Select with Feature Polygon" is available from the Map module. If a feature polygon is defined, the user can select nodes or elements in the mesh module or vertices in the data module that are inside or outside of the feature polygon.

Select By Material Type

The Select By Material Type... command selects all items of the current selection tool of a specified material. This command opens a dialog with a list of the defined materials and waits for the user to select a material type. This enables all nodes or elements that reference a specific material to be selected together.

Select By Dataset Value

The Select By Dataset Value... command opens a dialog to get a range from the user. All entities (nodes, elements, scatter points, etc.) of the current selection tool type whose scalar dataset value lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

Select By Area

The Select By Area... command opens a dialog to get a range from the user. All polygons whose area lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

Select By Length

The Select By Length... command opens a dialog to get a range from the user. All arcs whose length lies inside that range are selected. This enables all entities above or below threshold to be selected together for quick editing.

Confirm Deletions

By default, whenever a set of selected objects is about to be deleted, the user is prompted to confirm the deletion. This helps ensure that objects are not deleted accidentally. Selecting the Confirm Deletions command toggles this request for confirmation. When the option is off, the check mark next to the Confirm Deletions line in the menu disappears.

Current Coordinates

This command allows the user to tell SMS what coordinate system the data is reference with respect to. SMS supports several different global systems as well as a user defined local system.

Coordinate Conversions

The user converts the current data from on coordinate system to another. For more information, see the Coordinate Conversions article.

Single Point Conversion

This command opens a dialog which acts as a stand-alone coordinate converter. The user specifies a to and from coordinate system and a location. The location is converted to the new system within the dialog.

Time Settings

This command opens the Time Settings dialog. For more information, see the Time Settings article.

Materials Data

See the Materials Data article.

Project Metadata

This command allows the user to define metadata for the project. This documents a history of the project.

Copy to Clipboard

Copies the contents of the graphics window to the windows clipboard. This allows graphics to be easily transferred to documents and presentations.

Paste

Opens the Import Wizard with the contents of the windows clipboard. This requires that the contents be text values This allows graphics to be easily transferred to documents and presentations.

Preferences

This command allows the user to set program preferences. For more information, see the article Preferences.

Display Menu

The Display menu is the third standard menu available in all modules. The commands in the Display menu are used to control what entities are displayed and the attributes of those entities. The commands include:

Display Options

See the Display Options article.

Lighting Options

See the Lighting Options article.

 Refresh

When editing the image in the Graphics Window it occasionally becomes necessary to refresh the screen by redrawing the image. By default, SMS automatically updates the display when it is required (see Automatic Refresh below). To force the display to update, select the Refresh command from the Display menu or click the Refresh button. The process of redrawing can be aborted by pressing the Esc key.

Frame Image

Selecting the Frame Image command centers displayed data. This command adjusts the window boundaries so that all visible objects fit in the Graphics Window.

View

The view command brings up a sub menu. Items in the View submenu include:

- **View Angle –** set the bearing and dip of the look from direction. The bearing and dip values correspond to a rotation about the z and x axes. The bearing affects the horizontal angle (rotating the object in the xy plane), and the dip changes the vertical angle (shifting the viewing angle on the object to a higher or lower perspective). The object cannot be tilted sideways. Using only two viewing angles rather than three limits the viewing angles, but it is simpler and more intuitive.Plan view is a bearing of 0 degrees and a dip of 90 degrees.
- **Window Bounds –** The numerical model resides in a virtual world. The extents of that world displayed in the Graphics Window are the window boundaries. These boundaries can be altered using the Pan and Zoom tools. Alternatively, it is possible to precisely control the visible region by using the Set Window Boundaries command. The Set Window Boundaries dialog box appears, and the x and y limits of the viewing area can be set.
- **Plan –** Change the view in the Graphics Window to a plan view.
- **Front –** Change the view in the Graphics Window to a front view.
- **Oblique –** Change the view in the Graphics Window to a oblique view.
- **Side –** Change the view in the Graphics Window to a side view.
- **Previous –** Change the view to the previous view (the view before zooming and framing).

Plot Wizard

Opens the plot wizard. Details of how plots are generated and controlled are defined in the visualization tools. See the Plot Window article.

Plot Data

Edit the data plotted in the active plot. Make a plot active by clicking on it.

Plot Display Options

Change the display options for the active plot. Make a plot active by clicking on it.

Web Menu

The Web menu is one of the standard menus available regardless of the current module and model. The Web menu includes the following commands:

Import from Web...

Opens the web services utility which allows for the automated download and import of certain data types from the internet.

Find Data

This includes options to open to the Geo-Spatial Data Acquisition page on the XMS wiki. This page provides links to websites which provide various types of data which can be used in SMS.

Window Menu

The Window Menu is one of the standard menus available regardless of the current module and model. The Window Menu includes the following commands:

Cascade

Arranges all windows in an overlapping fashion within the SMS Graphics window.

Tile

Arranges all windows as non-overlapping vertical tiles within the SMS Graphics window.

Tile Horizontally

Arranges all windows non-overlapping horizontal tiles within the SMS Graphics window.

Active Window

A list of the currently open graphics and plot windows is shown at the bottom of the Window Menu. A check mark appears in front of the active window. Choose a window from the list to make it active.

Help Menu

The Help Menu is one of the standard menus available regardless of the current module and model. The Help Menu includes the following commands:

Help

Launches the Help File.

About

Displays the Version, Build Date, Contact Information, etc.

Related Topics

Menu Bar

Static Tools

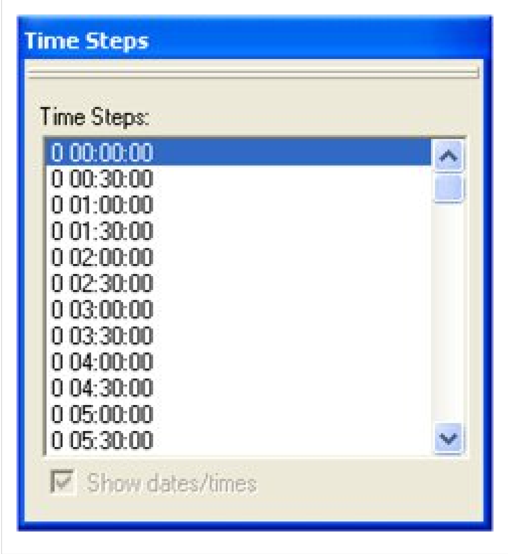
The Static Toolbar contains tools which are available in every module. These tools are tools for basic operations such as panning and zooming. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Static Tool palette.

Tool	Tool Name	Description
	Pan	The Pan tool is used to pan the viewing area of the Graphics Window. Panning can be done in 3 ways: • When the Pan tool is active, holding down the main mouse button while dragging moves the view. • If another tool is active and you don't want to switch tools, you can pan by holding down the F2 key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can hold it down and drag to pan the view.
	Zoom	The viewing area can be magnified/shrunk using the Zoom tool. Zooming can be done in the following ways: • With the zoom tool selected, clicking on the screen zooms the display in around the point by a factor of two. Holding down the shift key zooms out. • With the zoom tool selected, a rectangle can be dragged around a portion of the display to zoom in on that region. Holding down the shift key zooms out. • If another tool is active and you don't want to switch tools, you can zoom by holding down the F3 key and clicking and dragging with the mouse. • If your mouse has a middle button (or a mouse wheel), you can scroll the wheel to zoom in and out.
	Rotate	The Rotate tool provides a quick way to rotate the viewing location. Rotating can be done in the following ways: • With the rotate tool selected, holding down the mouse button and dragging the cursor in the Graphics Window rotates the object in the direction specified. A horizontal movement rotates the image about the z axis. A vertical movement rotates the image about the x and y axis. The amount of rotation depends on the distance the cursor moves while the mouse button is down. • If another tool is active and you don't want to switch tools, you can rotate by holding down the F4 key and clicking and dragging with the mouse. • The viewing angle can also be entered directly through the Display Options Dialog (General Options, View Tab).

Related Topics

- Layout of the Graphical Interface

Time Step Window



The *Time Steps Window* is used to select a time step to be active and is only visible if a transient dataset has been loaded into the project.

The *Time Step Window* is located below the Project Explorer by default, but it can be moved to anywhere on the window since it is a "dockable" toolbar. The *Time Step Window* can be resized by clicking on the window borders and dragging them. The Time Step Window only appears when a transient dataset is selected in the *Project Explorer*.

The display of time values in the *Time Step Window* is controlled by the settings in the Time Settings and Preferences dialogs.

Time Step Window Right Click Menu

Right clicking on the *Time Step Window* will give bring up the following options:

- Time Settings – Allows the user to change the how time is displayed.
- Time Preferences – Opens the *SMS Preferences* dialog.

Related Topics

- Layout of the Graphical Interface

Toolbars

There are several toolbars that can be displayed.

Macros

Many of the more frequently used menu commands can be accessed through the macro buttons. These buttons essentially serve as shortcuts to menu commands. Macro Toolbars include:

- Optional Macro Toolbar
- File Toolbar
- Display Toolbar

For more information, see the Macros article.

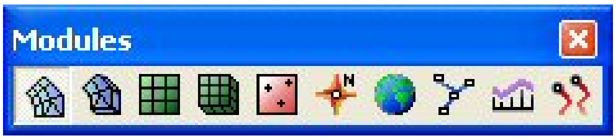
Static Toolbar

The Static Toolbar contains the tools which are available in every module. These tools are tools for basic operations such as panning and zooming. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. For more information, see the Static Tools article.

Dynamic Toolbar

When the active module is changed, the tools in the Dynamic Tool Palette change to the set of tools associated with the selected object/module. Each module has a separate set of tools. For more information, see the Dynamic Tools article.

Module Toolbar



Module Toolbar

The Module Toolbar is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a 2D finite element mesh) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

More Info...

Data Toolbar

The Data Toolbar is used to query objects displayed in the graphics window. For more information, see the Data Toolbar article.

Related Links

- Layout of the Graphical Interface

1.3. Support

Support

Support Forum

For news, updates or to post questions and participate in discussion topics for GMS, SMS, and WMS visit the Aquaveo support forum [1]. A weekly email summary can be requested by forum subscribers.

Related Topics

- GMS [2]
- SMS [3]
- WMS [4]

External Links

- Aquaveo Technical Support [5]

References

[1] http://forum.aquaveo.com/
[2] http://www.aquaveo.com/gms/
[3] http://www.aquaveo.com/sms/
[4] http://www.aquaveo.com/wms/
[5] http://www.aquaveo.com/technical-support/

Downloads

Aquaveo Download Page

- Aquaveo download page [1]

Related Topics

- Installing and Setting up SMS
- System Requirements
- License Agreement

References

[1] http://www.aquaveo.com/downloads

FTP Site Info

Return to Main Page

To upload files:

1.Zip them up

2.Go to this link.

| Upload Link [1]

3.Paste files in window

References

[1] ftp://pubftp.ems-i.com/upload/

License Agreement

AQUAVEO SOFTWARE LICENSE AGREEMENT

We understand that neither Aquaveo LLC nor its employees makes any warranty express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of the computer programs and documents herein ordered. We acknowledge that Aquaveo’s liability for damages associated with these programs is limited to the amounts paid to Aquaveo. We understand that we may register the executable on only one machine per license purchased. We also agree to not make the program available to more persons, at any given time, than the number of licenses purchased. We agree to not distribute this program in unmodified or modified form, outside our organization without the written permission of Aquaveo.

License agreements for software used by Aquaveo software

FTGL LICENSE AGREEMENT

http://ftgl.wiki.sourceforge.net/states: "FTGL is free software. You may use it, modify it and redistribute it under the terms of the MIT license or the GNU LGPL, at your option."

The MIT license link is to http://en.wikipedia.org/wiki/MIT_License, which states:

Copyright (c) <year> <copyright holders> Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Registering SMS

After installing SMS, it will need to be registered. Registration can be done with a password or with a hardware lock. When SMS is first launched, a dialogue box appears that has two options. The first button, **Demo Mode**, allows you to run SMS in demo mode. The second, **Enable** is used to enable the program. This is described below.

Password

If you are using a password to enable SMS, you will need to send information to your vendor about your machine to get your password. There are several ways you can send this information.

Register SMS with a Password [1]

1. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
2. Select *License code* for the Licensing method and enter the 7 digit alpha-numeric code that begins with the letter P. Click the **Next >** button.
3. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
4. The *Register SMS* dialog displays the registered components, licensing method, and license expiration dates.

Hardware Locks

Follow the instructions you received with the hardware lock to install the hardware lock and accompanying drivers. If you did not receive hardware lock instructions, or they have been misplaced, they can be found in the \Utils\Hwlock\Instructions directory on the CD. There are separate files for single user and network hardware locks. These files can be read using your web browser.

Demo Mode

If no valid license is detected, SMS runs in Demo Mode. All features of the software are enabled except printing, saving, and running models. This mode is intended to allow users to evaluate the software before making a purchase. Datasets, grids, or meshes can be read in, manipulated and viewed.

Evaluation Version

An evaluation version that is valid for 30-60 days may be requested by selecting the Evaluation button. You will be connected to a web utility, and a valid registration code will be sent to you via email. Once you receive the registration code, enter it into the dialogue box, and select register. Once you have evaluated SMS, please contact your vendor with any questions or to purchase.

Related Topics

- SMS System Requirements
- Hardware Locks

References

[1] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_Password_I.pdf

Hardware Locks

Single User USB Hardware Lock [1]

1. Install the hardware lock drivers. If hardware lock drivers have already been installed, skip to the next step. The drivers can be installed by running the **Sentinel System Driver Installer.exe** program found in the SMS installation directory.
2. After installing the hardware lock drivers, plug the Aquaveo hardware lock into an available USB port.
3. Start SMS and the program should automatically detect the hardware lock. If the Welcome dialog appears, click the **Register…** button.
4. Select **Hardware lock** for the Licensing method and click the **Next >** button.
5. In the Hardware lock options, select **Get license from a single user lock** and click the **Next >** button.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
7. The *Register SMS* dialog displays the registered components, licensing method, and license expiration dates.

Update a Single User USB Hardware Lock [2]

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and SMS installed.
2. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
3. Select **Hardware lock** for the Licensing method and click the **Next >** button.
4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.
5. In the *Hardware Lock* dialog, click the **Next >** button to burn the hardware lock.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*.
7. The *Register SMS* dialog indicates the registered components, the licensing method, and the license expiration dates.

Setup a Network License Server [3]

1. Install the Sentinel installation program that includes hardware lock drivers and the Sentinel Protection Server software. The installation program can be downloaded at www.aquaveo.com/downloads [1].
2. In the Sentinel installation wizard, select "Complete" for the setup type.
3. After installing the Sentinel lock drivers and server software, plug the Aquaveo hardware lock into an available USB port.
4. Ensure that the computer with the network hardware lock can be seen by other computers on the local network. Client machines can connect to the server by hostname or IP address.
5. The License Server is now ready to provide SMS licenses to client machines. Refer to the instructions for registering SMS with a Network USB Hardware Lock for more information.

Network USB Hardware Lock [4]

1. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
2. Select **Hardware lock** for the Licensing method and click the **Next >** button.
3. In the *Hardware lock* options, select **Get license from a network lock** and click the **Next >** button.
4. Enter the *IP address* or *Host name* of the server hosting the network hardware lock.
5. Click the **Browse Lock Setting...** button. This opens a web browser and tests the a connection to the hardware lock over a local network.
6. Click the **Apply Lock Setting...** button.
7. Once the "Lock license acquired" message appears, click the **Finish** button.
8. The *Register SMS* dialog displays the registered components, licensing method, hardware lock serial number, and license expiration dates.

Update a Network USB Hardware Lock [5]

1. Plug the Aquaveo hardware lock into a computer with hardware lock drivers and SMS installed.
2. Start SMS and select the **Register…** button when the welcome screen appears. If the welcome screen does not appear automatically, select **Register...** from the *Help* menu in SMS.
3. Select **Hardware lock** for the Licensing method and click the **Next >** button.
4. In the *Hardware lock* options, select **Modify lock on this computer with the following code** and click the **Next >** button.
5. In the *Hardware Lock* dialog, enter the number of licenses to burn on the lock. This number is typically the same as the number of licenses purchased and available. Click the **Next >** button to burn the hardware lock.
6. If the registration is successful, click **Finish** to exit the *Registration Wizard*. Please note that the *Registration* dialog may not show the enabled components. To verify the enabled components on the network lock, refer to the instructions for registering SMS with a Network Lock.
7. Return the network hardware lock to the computer serving as the license server if necessary.

Related Topic

- Registering SMS

References

[1] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_SUHWL_I.pdf
[2] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_Burn_SUHWL_I.pdf
[3] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_NW_License_Server_Setup_I.pdf
[4] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_with_a_NWHWL_Client_I.pdf
[5] http://www.aquaveo.com/pdf/license_instructions/v1/Register_SMS_Burn_NWHWL_I.pdf

Graphics Card Troubleshooting

XMS (WMS, GMS, or SMS) use OpenGL for rendering graphics. OpenGL is a graphics standard, but each implementation is maintained by individual graphics card companies. Different graphics cards and drivers support different versions of the OpenGL standard. XMS currently uses features up to version 1.5 of OpenGL (as of April 2009 version 3.1 was most recent version).

Some graphics cards, as well as remote desktop, do not support functionality through OpenGL version 1.5. This is mostly a problem with older integrated graphics cards, in particular those manufactured by Intel. This page will give you some ideas on troubleshooting these problems. The best solution is to get a graphics card that supports later versions of OpenGL. You will see improved performance as well as be able to access all the features of XMS.

Remote Desktop

XMS (WMS, GMS, or SMS) will have reduced capability when running remote desktop.

Since remote desktop only supports OpenGL version 1.1 not all of the features of XMS may be available.

1. One solution is to use a different remote control software that utilizes the graphics card of the computer you are controlling. www.logmein.com [1] has free and paid versions of remote desktop that behave better with XMS. RealVNC is a program that does this and can be purchased at a reasonable cost. There is a free version but it has not been tested with the XMS software. See VNC Homepage [2] for more information.
2. Another solution is to use the Mesa software rendering option available in the application's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

Parallels Desktop for Mac

XMS has reduced capability when running in a pure virtual PC through Parallels Desktop for Mac. Although Parallels version 6.0 provides OpenGL version 2.1 support (instead of OpenGL version 1.1) when "Enable 3D acceleration" is selected in the virtual machine's hardware configuration, the Parallels virtual video card adapter does not render all XMS graphics correctly. The solution is to use the Mesa software rendering option available in XMS's graphic preferences. See the section below on OpenGL Graphics Dialogs for discussion of this option.

If you are running XMS in a virtual PC utilizing a Boot Camp partition then Parallels uses the actual graphics card installed in the Mac. See sections below regarding graphics card issues.

OpenGL Graphics Dialogs

XMS (post WMS 8.2, GMS 7.0 onward, and SMS 10.1 onward) have dialogs that allow the selection of OpenGL support. The choice is between the system default library and the Mesa software library. The system default can change based upon current conditions such as a remote login. Not all system defaults support all needed graphics functionality. Therefore Mesa is provided for better functionality at a potential reduction in speed. However, Mesa may produce poor images when printing. The user can make this tradeoff in the graphics dialog found in preferences. The dialog provides 4 options so that on subsequent runs XMS will:

1. Ask which graphics library to use if the system does not support all OpenGL functionality needed by XMS. This option is initially set and gives the following options:
   1. Autoselect the Mesa software library for this run if the system default does not support all functionality. XMS will not prompt on subsequent runs. It will just check support and select a library.
   2. Use the system default library on this run (and on future runs if the "Do not ask again box" is checked).
   3. Use the Mesa software library on this run (and on future runs if the "Do not ask again box" is checked).
2. Autoselect the Mesa software library if the system default does not support all functionality.

3. Always use the system default library.
4. Always use the Mesa software library.

Determining Graphics Card Manufacturer

Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. You can use a simple diagnostic program called dxdiag [3] to determine your computer's hardware, operating system, and graphics card. To use the dxdiag [3] program:

1. Select "Start"
2. Choose "Run."
3. Type "dxdiag" in the box and click "OK."
4. Click "Yes" to the prompt, and the program will begin running.
5. Select the "Display" tab and the Name listed under the "Device" section is the name of your graphics card.

You can also:

1. Right-click on the desktop and select "Properties"
2. In the Display Properties dialog, click on the "Settings" tab
3. Your video card manufacturer and chipset is shown below the "Display:" line
4. Look for the names NVIDIA, ATI, Intel, Matrox, SiS, S3, etc.

Updating Laptop Graphics Card Drivers

If you have a laptop, visit the laptop manufacturer's website (Dell [4], HP or Compaq [5], Toshiba [6], Sony [7], etc.) to get the most recent driver.

Updating Desktop Graphics Card Drivers

If you are using a desktop computer, visit the graphics card manufacturer's website to download the latest driver. Listed below are a few common graphics cards and links to download their drivers:

- 3DLabs [8]
- ATI [9]
- Diamond [10]
- Elsa [11]
- Intel [12]
- Matrox [13]
- nVidia [14]
- S3 [15] – Not all S3 card support OpenGL 1.5 which is required for all display options to be enabled.
- SIS [16] – Not all SIS card support OpenGL 1.5 which is required for all display options to be enabled.
- VIA [17] – Not all VIA card support OpenGL 1.5 which is required for all display options to be enabled.

Updating Windows Operating System

Many problems are resolved by keeping the windows operating system and hardware drivers up to date using the windows update site [18]. Hardware updates are often only installed if the "Custom" or "Optional" updates are included.

Updating XMS Software

Many problems are resolved by installing the latest version of XMS. Bugfixes and updates are released frequently. The updates can be downloaded at the Aquaveo Download Center [1].

Known Graphics Issues

- Issue: Graphic symbols are not displayed correctly and sometimes corrupt text lines located next to them.

  Hardware: Make: ATI Technologies Inc. Model: RADEON X600 PRO (0x5B62) Name: ATI Radeon X300/X550/X1050 Series

  Solution: Updating the driver will allow the symbols to display correctly, but the text corruption still remains.

Switch from Hardware to Software Rendering

THE FOLLOWING SHOULD BE ATTEMPTED ONLY IF THE OTHER SOLUTIONS PRESENTED DO NOT RESOLVE THE DISPLAY ISSUES

If you have updated your graphics driver and are still having problems, you can download this opengl32.dll ZIP file [19] and unzip the "OpenGL32.dll" and the "Glu32.dll" file to the directory where XMS is installed. Close and re-open XMS so this DLL is used for displaying XMS objects. Placing these DLL's in your XMS directory will fix most graphics-related issues, such as problems with displaying triangles on large TIN or DTM datasets and other problems with displaying large amounts of data. The following are known disadvantages to using this DLL for displaying:

- Displaying graphics using this DLL will likely be slower since software is used to display your graphics instead of your computer's graphics hardware. Panning, zooming, and rotating operations will be significantly slower.
- Some entities, such as symbols, are currently not displayed correctly when using this DLL. Only squares and circles will be displayed. Changing all symbol display options to squares or symbols will allow you to work around this problem. We are currently working on trying to fix this problem of symbols not displaying when using this DLL. (THIS PROBLEM HAS NOW BEEN FIXED IN SOME BETA VERSIONS OF XMS COMPILED AFTER March 31, 2009) In general, you will not want to use this DLL unless you are working with large datasets that have display issues where XMS closes unexpectedly.

Contacting Support

If you continue to experience problems after updating your graphics card drivers, contact support [5].

References

[1] http://www.logmein.com/
[2] http://www.realvnc.com/
[3] http://en.wikipedia.org/wiki/DxDiag
[4] http://www.dell.com/
[5] http://welcome.hp.com/country/us/en/support.html
[6] http://www.toshiba.com
[7] http://www.sony.com
[8] http://www.3dlabs.com/support/drivers/
[9] http://ati.amd.com/support/driver.html
[10] http://www.diamondmm.com/
[11] http://www.elsa.com/supports/download.asp
[12] http://support.intel.com/support/graphics
[13] http://www.matrox.com/mga/support/drivers/latest/home.cfm
[14] http://www.nvidia.com/content/drivers/drivers.asp
[15] http://www.s3graphics.com/drivers.jsp
[16] http://www.sis.com/support/support_prodid.htm
[17] http://www.viaarena.com/default.aspx?PageID=2
[18] http://update.microsoft.com
[19] http://wms.aquaveo.com/OPENGL32.zip

Setting up SMS – Introduction

Installing

The installation wizard will guide you through the installation process. You will have the option to install different parts of the SMS program including the executable files, tutorial files, documentation files, etc. If you are missing a part of the installation, reinstall and verify that all parts are selected to be installed.

Registering

See Registering SMS for information regarding registering SMS using a hardware lock or password.

Program Defaults

When you start SMS there are default values set for directories, display options, etc. Each project also saves settings associated with the project. As a project is read, the values are set to the project settings. The default settings only appear when you are creating new projects. You can modify the default settings for a new project by choosing Save Settings from the File menu. This will replace the default settings with your current settings.

Frequently Asked Questions

Q: My hardware lock doesn't work.

A: Visit the SMS:Hardware Locks page for Hardware lock troubleshooting. Review the hardware lock troubleshooting guides: Single User Locks [1] and Network Locks [2].

Q: I encountered an error when trying to install SMS.

A: The most common cause for installation errors is running the installation program without "Administrator" priviledges. Also check that the installation diectory is a valid location and that it isn't "read-only."

Q: Where can I get the latest build updates of SMS?

A: See Downloads or visit the Aquaveo download page [1].

Q: What is "Demo Mode?"

A: SMS runs in Demo Mode if a valid license is not present. In Demo Mode, printing and saving are disabled.

Q: I purchased SMS. How do I enable the software and get out of Demo Mode?

A: SMS can be enabled with a password or hardware lock. To obtain a password or hardware lock, contact your software vendor. Passwords enable a single version of SMS on a single machine. Passwords are machine specific. When obtaining a password, you will need to provide your vendor with your computer's register string. The register string is listed in the Register dialog (File | Register.) Hardware locks enable a roaming license of SMS. In order to enable SMS with a hardware lock, the lock must be attached to the machine when running SMS. See Registering SMS for more information.

Q: In the graphics window, letters and numbers appear instead of points and nodes. How can that be fixed?

A: If letters and numbers appear in the graphics window instead of regular points and nodes, then there was an error installing the SMS font or the font was corrupted. Usually this problem clears up when the computer is restarted after the SMS installation process is finished.

Q: My password doesn't work.

A: Passwords are not case sensitive but register strings **ARE** case sensitive. Double check the password data sent to you by your vendor and make sure the register is correct and you have entered the correct password.

Q: What does the error "This application has failed to start because MSVCR71.dll was not found" mean?

A: Your machine is missing the file MSVCR71.dll. This file should have been installed on your computer with the Windows Operating System. To fix this, download and reinstall MSVCR71.dll. MSVCR71.dll is available for download on many websites. Use a search engine, such as Google, to find it.

Related Topics

- Downloads
- System Requirements
- License Agreement

References

[1] http://ems-i.com/Support/single_user_hardware_locks.html
[2] http://ems-i.com/Support/network_hardware_locks.html

System Requirements

System requirements for GMS, SMS and WMS.

Windows 7[1]

Windows 7 is supported in GMS 7.0, SMS 10.1, WMS 8.3 and greater versions only.

Component	Minimum Required	Recommended
RAM	1 GB	4 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	We recommend nVidia Quadro based graphics cards [2] because of their superb OpenGL support. The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

Windows Vista[3]

Windows Vista is supported in GMS 7.0, SMS 10.0, WMS 8.1 and greater versions only.

Component	Minimum Required	Recommended
RAM	1 GB	4 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	We recommend nVidia Quadro based graphics cards [2] because of their superb OpenGL support. The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

Windows XP

Windows XP is recommended for GMS 7.0, SMS 10.0, WMS 8.1 and greater versions.

Component	Minimum Required	Recommended
RAM	512 MB	2 GB or greater
CPU	XMS software is CPU intensive. We recommend the fastest CPU your budget allows.	
Hard Disk Free Space	300 MB	300 MB or greater
Graphics Card	**For all display features to be enabled, OpenGL 1.5 must be supported.**	We recommend nVidia Quadro based graphics cards [2] because of their superb OpenGL support. The use of a dedicated graphics card is strongly recommended. Integrated graphics cards are often problematic.
Minimum Resolution	1024x768	1024x768 or greater

[1] Windows 7 is supported in GMS 7.0, SMS 10.1 and greater versions only.
[2] http://en.wikipedia.org/wiki/NVIDIA_Quadro
[3] Windows Vista is supported in GMS 7.0, SMS 10.0, WMS 8.1 and greater versions only.

Notes

- You may have display problems when running over remote desktop. This can usually be fixed by restarting the software after beginning/ending a remote desktop session. Remote Desktop cannot be used with single-user/standalone locks, only with network locks.
- Always download and install the latest drivers from your graphics card vendor. Graphics card problems are often due to using the wrong or outdated drivers. See Graphics Card Troubleshooting for instructions on how to download and install graphics card drivers. If you continue to experience problems after updating your graphics card drivers, contact support (http://www.aquaveo.com/technical-support/).

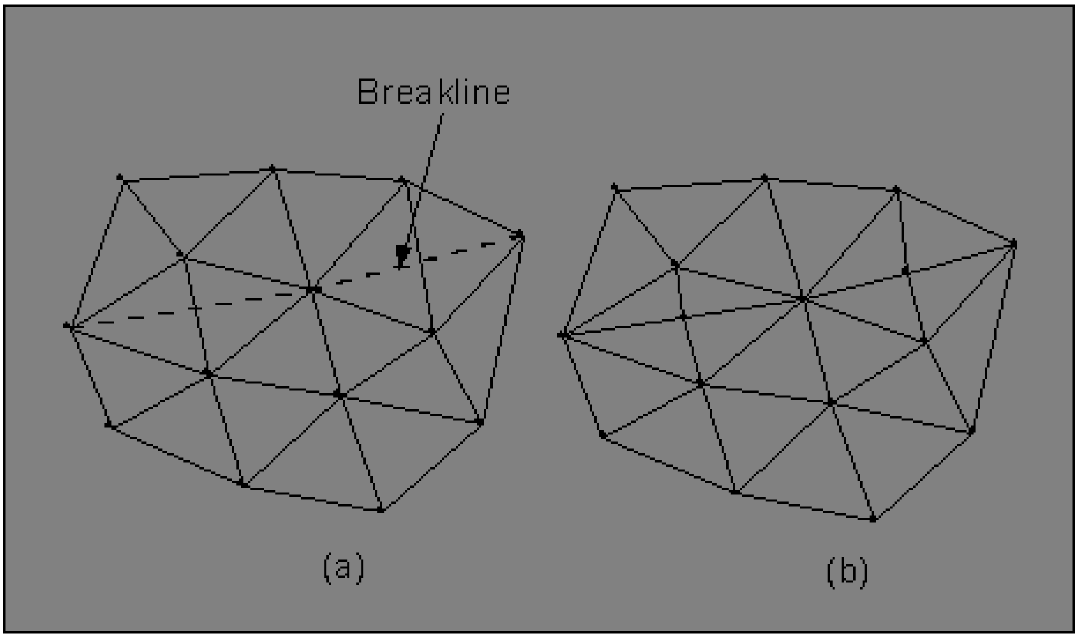
External Links

- Description of the DirectX Diagnostic Tool (http://support.microsoft.com/kb/190900)

2. Functionalities

Breaklines

A breakline is a feature or polyline representing a ridge, thalweg, or other shape that the user wishes to preserve in a surface made up of triangular elements or scatter set. In other words, a breakline is a series of edges to which the mesh or scatter triangles should conform to, i.e., not intersect.



Breaklines (a) Original Triangulation and Breakline. (b) Triangulation After the Breakline has been Processed by inserting new mesh nodes along the breakline.

Mesh Module Breaklines

Breaklines are processed using the Force Breaklines command from the Nodestrings menu. How breaklines are processed is controlled by the breakline options in the Nodestring Options Dialog.

Scatter Module Breaklines

Breaklines are processed using the Force Breaklines command from the Breaklines menu. Scatter breaklines are always processed by swapping triangle edges to ensure that the edges of the triangles will conform to the breakline.

Importing Scatter Breaklines

Scatter Breaklines can be imported along with scatter data using the file import wizard. In order to import breaklines, the tabular file must be prepared in one of two supported formats. They are illustrated below. In either case, an additional column of data defines the breakline information. In the import wizard, this column should be mapped as "Breakline". This brings up the Scatter Breakline Options Dialog

Note: Scatter breaklines must be imported at the same time as their corresponding scatter vertices.

Example Files

Example of a tab delimited file using breakline names:

```
xcoord	ycoord	zcoord	name
215962.9	85203.098	1.483	Breakline1
215957.638	85193.069	1.483	Breakline1
215963.278	85184.35	1.483	Breakline1
215979.111	85179.328	1.483	Breakline1
216056.51	85209.371	1.483	Breakline1
215992.462	85201.477	7.034	Breakline2
216127.386	85264.681	7.034	Breakline2
216267.187	85327.936	7.034	Breakline2
216371.217	85381.431	7.034	Breakline2
219261.939	90247.944	8.763
219461.211	90220.556	9.167
219678.994	90179.064	9.468
```

Example of a tab delimited file using the following breakline tags:

- Start: 1
- Continue: 2
- End: 4
- Not in breakline: 5

```
xcoord	ycoord	zcoord	breakline_tag
215962.9	85203.098	1.483	1
215957.638	85193.069	1.483	2
215963.278	85184.35	1.483	2
215979.111	85179.328	1.483	2
216056.51	85209.371	1.483	4
215992.462	85201.477	7.034	1
216127.386	85264.681	7.034	2
216267.187	85327.936	7.034	2
216371.217	85381.431	7.034	4
219261.939	90247.944	8.763	5
219461.211	90220.556	9.167	5
219678.994	90179.064	9.468	5
```

Related Topics

- Editing 2D Meshes

Object Info

Information Dialog

The Get Info command reports basic information concerning the data type associated with the active module. Information is available for the following modules:

Cartesian Grid Module Information

The following information is shown on the Cartesian Grid Module tab of the Information Dialog:

- Number of cells
- Number of rows
- Number of columns
- Minimum Z value
- Maximum Z value
- Angle
- Cell size
- Number of monitoring stations
- Number of ocean cells
- Number of land cells

Map Module Information

The following information is shown on the Map Module tab of the Information Dialog:

- For all coverages:
  - Number of points
  - Number of nodes / vertices
  - Number of arcs
  - Number of polygons
- For selected coverage:
  - Number of points
  - Number of nodes / vertices
  - Number of arcs
  - Number of polygons

Mesh Module Information

The following information is shown on the Mesh Module tab of the Information Dialog:

- Maximum element front width
- Maximum node half band width
- Number of elements
- Maximum element ID
- Number of nodes
- Maximum node ID
- Minimum Z value
- Maximum Z value
- Element type
- Number of triangular elements

- Number of quadrilateral elements
- Model specific Info
  - RMA2
    - Transition elements
    - Junction elements
    - Control elements
    - Linear elements
    - 1D nodes without 1D
  - FESWMS
    - Number of culverts
    - Number of piers
    - Number of weirs
    - Number of drop inlets
    - Max ceiling value
    - Min ceiling value
  - Generic Mesh Model
    - Model name

Scatter Module Information

The following information is shown on the Scatter Module tab of the Information Dialog:

- For all scatter sets
  - Number of points
  - Number of triangles
- For selected scatter sets
  - Scatter set ID
  - Number of points
  - Number of triangles

Related Topics

- File Menu

Select/Delete Data...

Select/Delete Data... (Feature Objects Menu) The Trim Data command (Feature Objects menu, Map module) is available when one or more polygons are selected. The user may select or delete data that is located inside or outside of selected polygons. Options are provided to select or delete mesh node, elements or duplicate nodes, scatter points or triangle, Cartesian grid cells or cell location that are partially (triangles/elements that cross a boundary) or completely inside/outside the polygons.

Function Type

- **Select –** Select objects
- **Delete –** Delete objects

Data Domain

- **Inside polygons(s) –** Trim data inside selected polygons.
- **Outside polygon(s) –** Trim data outside selected polygons.
- **Treat boundary as [Outside | Inside]** - Treat data that lies on polygon boundaries as if it were outside/inside of the polygon(s).

Select Data Type

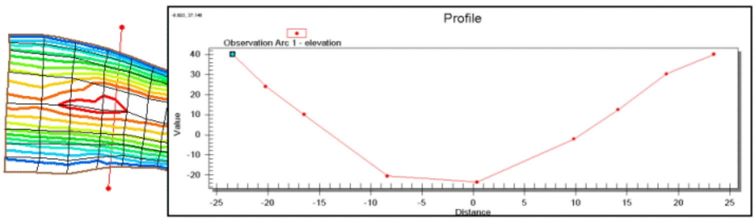
Choose the data type to select or delete

- **Mesh**
  - **Nodes** (Select) – Select mesh nodes only.
  - **Nodes and Elements** (Delete) – Trim mesh nodes and elements.
  - **Elements –** Select or delete mesh elements only.
- **Scatter**
  - **Points** (Select) – Select scatter points only
  - **Points and Triangles** (Delete) – Trim scatter points and triangles.
  - **Triangles –** Trim triangles only.
  - **Scatter Sets to Trim –** Click on a scatter set in the window to toggle it on/off for trimming. Push the Active Set button to only select the active scatter set and the All Sets button to turn on/off all sets. Points and/or triangles are deleted only from the selected sets.
- **Cartesian grid cells**
- **Cartesian grid cell locations –** this can be useful in working with elevation values in [[TUFLOW grids.

2.1. 2D Plots

Plot Window

At a glance



A plot showing a cross-section view of a river channel

- 2D plots to visualize results and compare to measured values
- Profile plots view scalar data along an arc
- Time-series plots view scalar, vector, or flux (flowrate) data at a point or across an arc
- Several kinds of plots can be used to compare model results with measured data

In SMS, the *Plot Window* can be used to display various plots. Plots aid the user in extracting data from two or three dimensional objects, model verification, and defining one-dimensional river models. By selecting *Display Options* | *Plot Wizard*, a user is given help in a step by step process to create a variety of plots. The list of currently available plots in the *Plot Wizard* is shown below. All of the plots listed below are associated with Observation Coverages, except the 1-dimensional river plots.

Available Plot Types

1. Computed vs. Observed Data
2. Residual vs. Observed Data
3. Error vs. Simulation
4. Error vs. Time Step
5. Error Summary
6. Time Series
7. Observation Profile
8. TUFLOW Cross-sections
9. PTM Gages
10. Runup/Overtopping Transect
11. Runup/Overtopping Solution
12. GenCade Inlet Time-series
13. GenCade Shoreline Change and Transport
14. Angle Representation Region (ARR) mesh quality assessment plot

Plot Wizard

The Plot Wizard is used to bring up a plot window which will display the specified plot. There are two steps that a user is directed through to create a plot.

Step 1 – User selects the desired plot type from the list. Some plot types are hidden unless certain requirements are met.

- The "TUFLOW Cross Section" plot requires a TUFLOW Cross-Section coverage.
- The "Runup/Overtopping Transect" and "Runup/Overtopping Solution" plots require a Runup/Overtopping simulation.
- The plot "PTM Gages" requires a PTM Gage coverage.
- The "GenCade Inlet TS" and "GenCade Shoreline" plots require a GenCade 1D grid.

Each plot is described as they are highlighted from the list. The Plot Wizard will also tell the user if they have appropriate data to be able to make certain plots.

Step 2 – The plot is defined by selecting what data will compared, which time step will be shown, and other pertinent information. Each plot's options are described in more detail by click on the Plot Type links above.

By default, a plot is displayed after being created in a separate display window than the simulation data. A plot window can be minimized, moved, and resized just like any other window.

Plot Options

Right clicking on a plot will bring up a menu of commands for formatting the data in the plot as well as giving access to tools for exporting the plot data for use in spread sheets or other plotting utilities.

Plot Window Right Click Menu

The following *Plot Window* right mouse click menus are available:

- Plot Data – Opens step 2 of the plot wizard
- Display Options – Opens the Profile Customization dialog
- Axis Titles – Opens the Axis Titles dialog
- Set as Display Defaults
- Legend – Set the legend location (Top, Bottom, Left, Right)
- Symbol Size – Set the symbol size (Micro, Small, Medium, Large)
- Frame Plot – If the view is zoomed in to a portion of the plot, resets zoom extents
- Maximize Plot – Makes the plot appear full screen
- View Values – Opens the View Values dialog
- Export/Print – Opens the Exporting Profile dialog

Time Settings Options

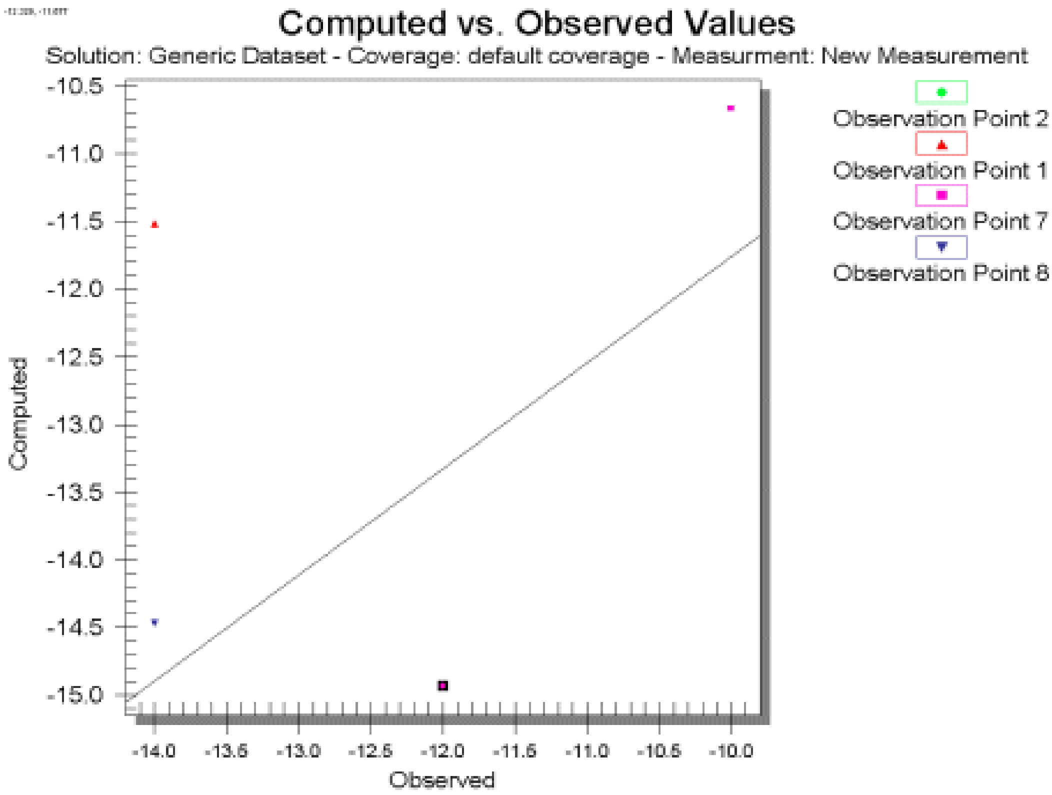
The formatting of Date / Times displayed in plots is controlled by the global time settings. See Time Settings for information on how to change the format times are displayed in.

Related Topics

- Visualization

Computed vs. Observed Data

A Computed vs. Observed plot is used to display how well the entire set of observed values for observation points matches the solution data. On this plot is drawn a 45 degree line, representing what would be a perfect correspondence between observed data and solution values. Then, one symbol is drawn for each observation point at the intersection of the observed and computed values for the point. This plot can show the trend of the solution values with regards to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the Plot Wizard by setting the plot type to Computed vs. Observed. A sample plot is shown in the figure.



Computed vs. Observed Plot Options

After the plot type is set in Step 1 of the Plot Wizard, the Next button is clicked to go to Step 2, which displays the following items.

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the Feature Objects | Attributes dialog, being plotted.

Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Error Summary

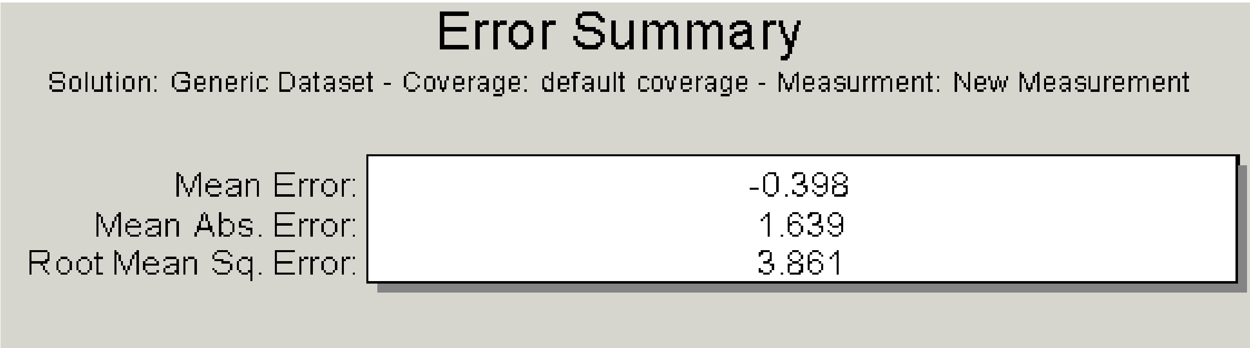
An Error Summary plot is used to display a text listing of the mean error, mean absolute error, and root mean squared error for a dataset and the observed values associated with a mesh or grid on observation points in the Observation Coverage. The errors shown are the mean errors for all observation points with computed data.

Mean Error – This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

Mean Absolute Average – This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.

Root Mean Square – This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

Error Summary plots are created in the Plot Wizard by setting the plot type to Error Summary. A sample plot is shown in the figure.



Error Summary Plot Options

After the plot type is set in the first step of the Plot Wizard, the Next button is clicked to move to the second step of the Plot Wizard.

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the Feature Objects | Attributes dialog, being plotted.

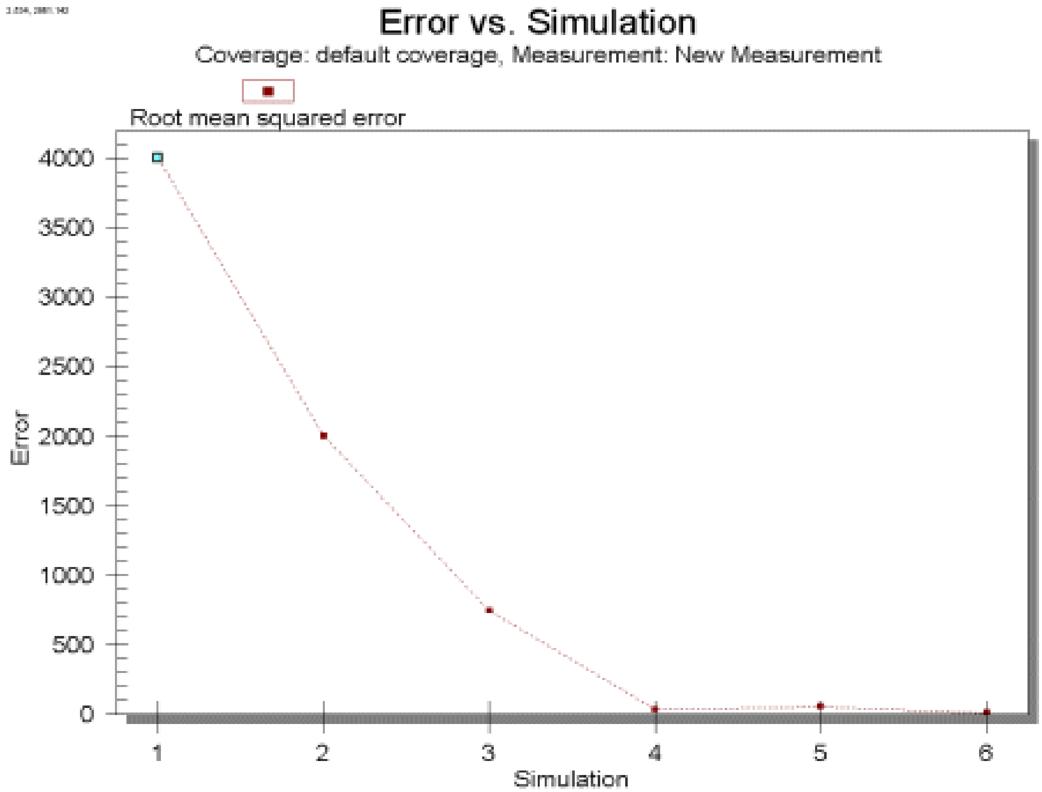
Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Error vs. Simulation Plot

An Error vs. Simulation plot is generally used with constant simulations and measurement types, although it may be used in transient simulations. This plot can display the mean error, mean absolute error, and root mean squared error between successive solutions and a set of observed data. Various simulations would be run after changing model parameters, such as material roughness values and/or eddy viscosities. The plot will show trends in the solution to see if model parameter changes are causing better calibration with measured field data. Error vs. Simulation plots are created in the Plot Wizard by setting the plot type to Error vs. Simulation. A sample plot is shown in the figure.



Error vs. Simulation Plot

After the plot type is set in the first step of the Plot Wizard, the second step of the Plot Wizard shows the options for the Error vs. Simulation plot.

Solutions – This box lists all available solutions for the simulation. Select the desired solution to use its datasets in the plot.

Move Up/Move Down – SMS initially shows the solutions in the order they were opened. However, this is not necessarily the order in which they were run. To change the order, highlight a solution and move it up or down to rearrange their order.

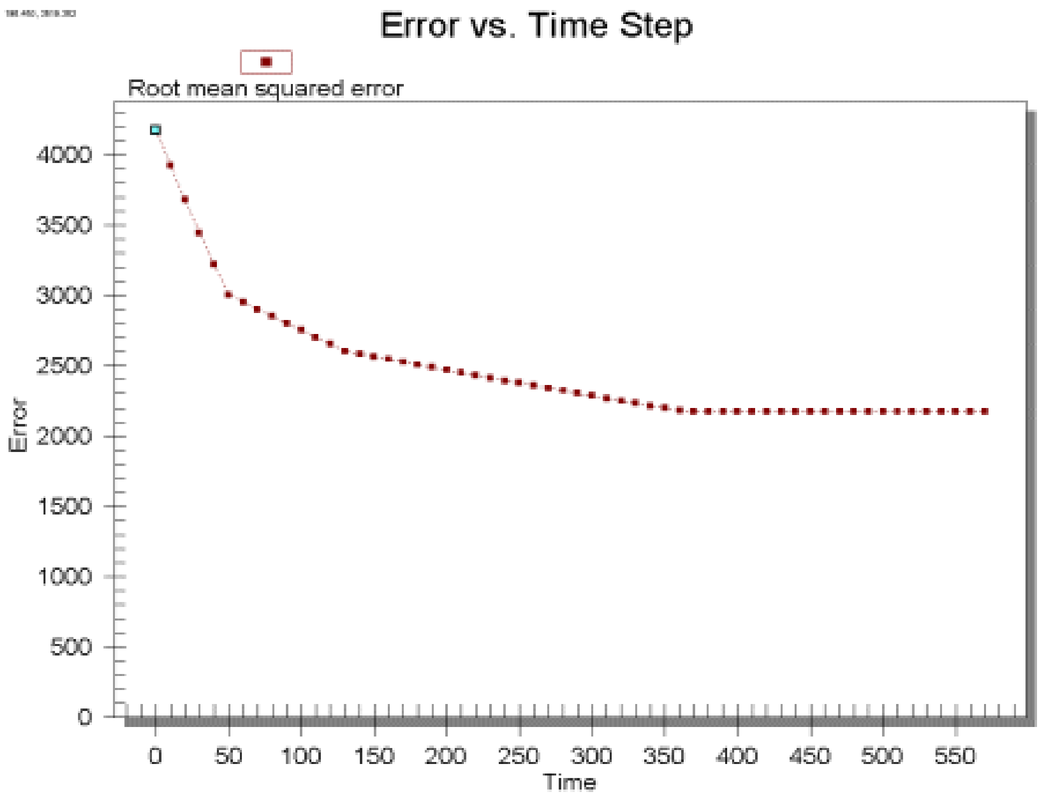
Check Box Options – There are three options that can be turned on or off. The three options determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Related Topics

- Plot Window

Error vs. Time Step Plot

An Error vs. Time Step plot is used with transient simulations to display the mean error, mean absolute error, and root mean squared error between a solution and observed data as a function of time. This plot is shown for a single Data Set of a mesh or grid as an average of all observation points assigned to the specified measurement type in the Observation Coverage. The measurement type should be defined as a transient measurement. Although this plot can be used for constant measurement types, only a single point will be shown in the plot, and you would be better off using the Error Summary Plot. Transient measurement types will show the average errors at each time step of the data set. Error vs. Time Step plots are created in the Plot Wizard by setting the plot type to Error vs. Time Step. A sample plot is shown in the figure.



Error vs. Time Step Plot Options dialog

After the plot type is set in the first step of the Plot Wizard, the plot options are shown in Step 2 of the Plot Wizard, contains the following options.

Computed – This lists all available datasets. The dataset to be analyzed should be chosen.

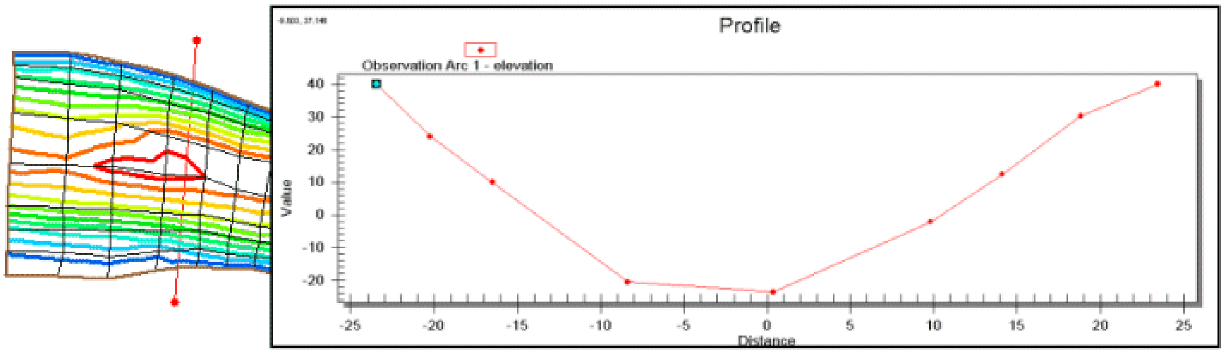
Check Box Options – There are three options that can be turned on or off. The three options determine whether the mean error, mean absolute error, and root mean squared error plots should be shown. Because these values are an average of all observation points, their line and symbol styles are not linked to any one observation point, but can be defined by clicking on the appropriate canvas window in the dialog.

Related Topics

- Plot Window

Observation Profile

A Profile plot is used to display the variation of one or more scalar datasets associated with a mesh or grid along observation arcs in the Observation Coverage. Profile plots are created in the Plot Wizard dialog by selecting Observation Profile from the plot type list. When an arc is selected two small arrows appear at either end of the arc. These arrows indicate the viewing direction for the plots. To change the viewing direction select the arc and execute the "Feature Objects | Reverse Arc Direction" command. A sample plot is shown in the figure.



Profile Plot Options

After the plot type is set in Step 1 of the Plot Wizard, the profile plot options need to be defined. The following options must be set for a profile plot:

Coverage

A profile plot operates on a single observation type coverage. The following coverage related options are available:

- **Coverage** – If multiple observation type coverages exist, the coverage to use for the profile plot must be selected.
- **Extraction method**
  - **Model Intersections** – Profile plot points are based on intersections of the specified feature arcs and element, cell, or triangle edges.
  - **Points and Vertices** – Profile plot points are interpolated at the location of points and vertices on the specified feature arcs.

Dataset

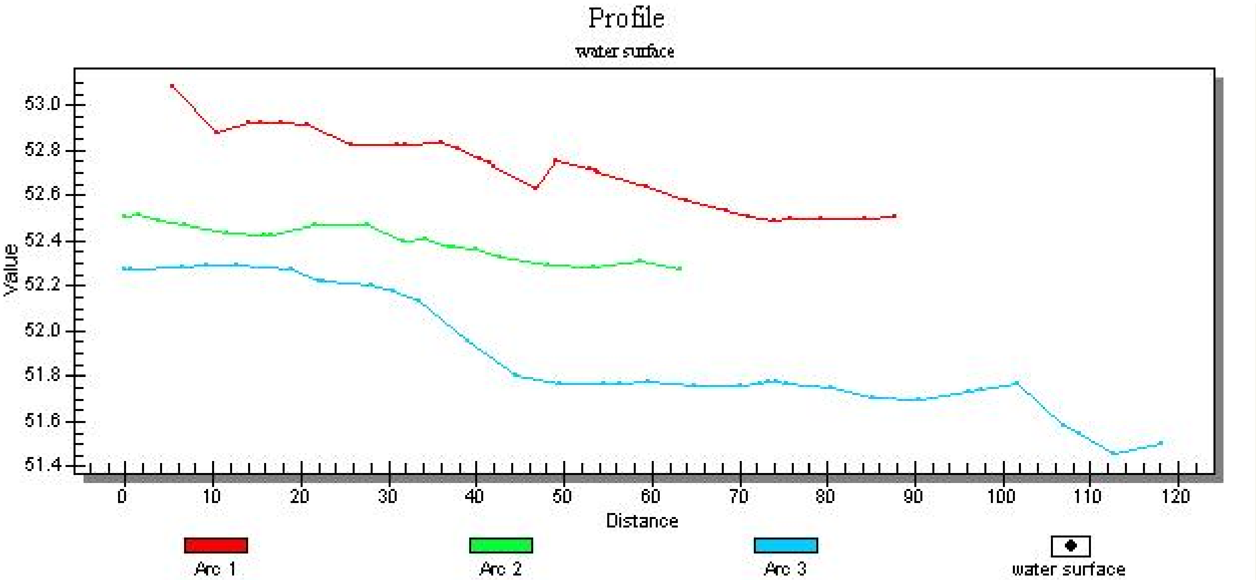
- **Active dataset** – Profile plot points are based on the active dataset. The profile plot will update when the active dataset is changed.
  - **Module** - Since each module contains an active dataset, when using the active dataset option, the module must be specified.
- **Specified dataset(s)** – Profile plot points are based on the specified dataset(s). Datasets from different modules can be specified.

Time step

- **Active time step –** Profile plot points are based on the active time step. The profile plot will update when the active time step is changed.
- **Specified time step –** Profile plot points are based on the specified time step.
- **Use active dataset and time step –** This option causes the plot to display the values of the active dataset and time step for each arc being plotted. When the active dataset changes, the plot is recomputed and updated.
- **Use selected dataset and time step –** This option causes the plot to display the values of one or more specified datasets or time steps for each arc being plotted. Changing the active dataset does not affect the plot. Check the check-box of the dataset that will be viewed from the list box.

Plotting With Multiple Arcs Selected

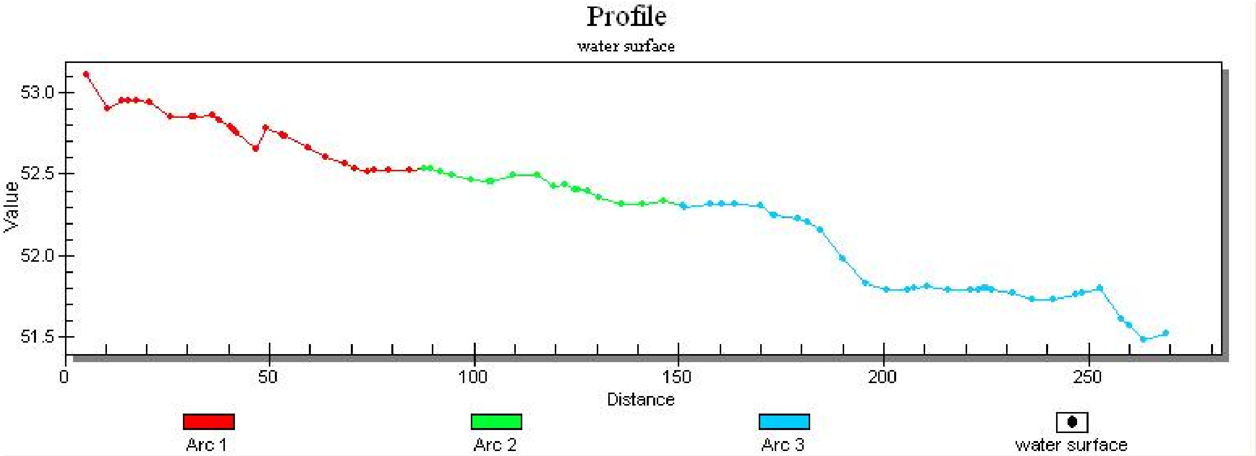
There are two ways in which an observation profile can be created when multiple arcs will be graphed. Multiple arcs can be graphed on a single plot was so they appear in separate segments as shown below.



Multiple observation arcs can also be plotted to look continuous if they are part of an arc group by following these steps:

- Create a profile arc that composed of more than one arc (this means the arcs must be connected at the end points and start points)
- Choose the menu command Feature Objects | Create Arc Group
- Create the observation profile plot like before by using the Plot Wizard

When the plot is generated, it will look as shown below where the line is continuous. The different arcs are drawn in their respective colors, but are linked together end to end.



Related Topics

- Plot Window

Profile Customization Dialog

The Profile Customization dialog allows you to edit many plot properties. The plot options are organized onto the following tabs in the dialog:

- **General Tab** – plot title, border style, viewing style, font size, numeric precision, and grid line style can be changed
- **Axis Tab** – contains x and y axis information
- **Font Tab** – plot font style can be edited
- **Color Tab** – any color option can be changed here
- **Export Button** – allows the user to export the plot and plot data in different file formats, to a printer, or the clipboard

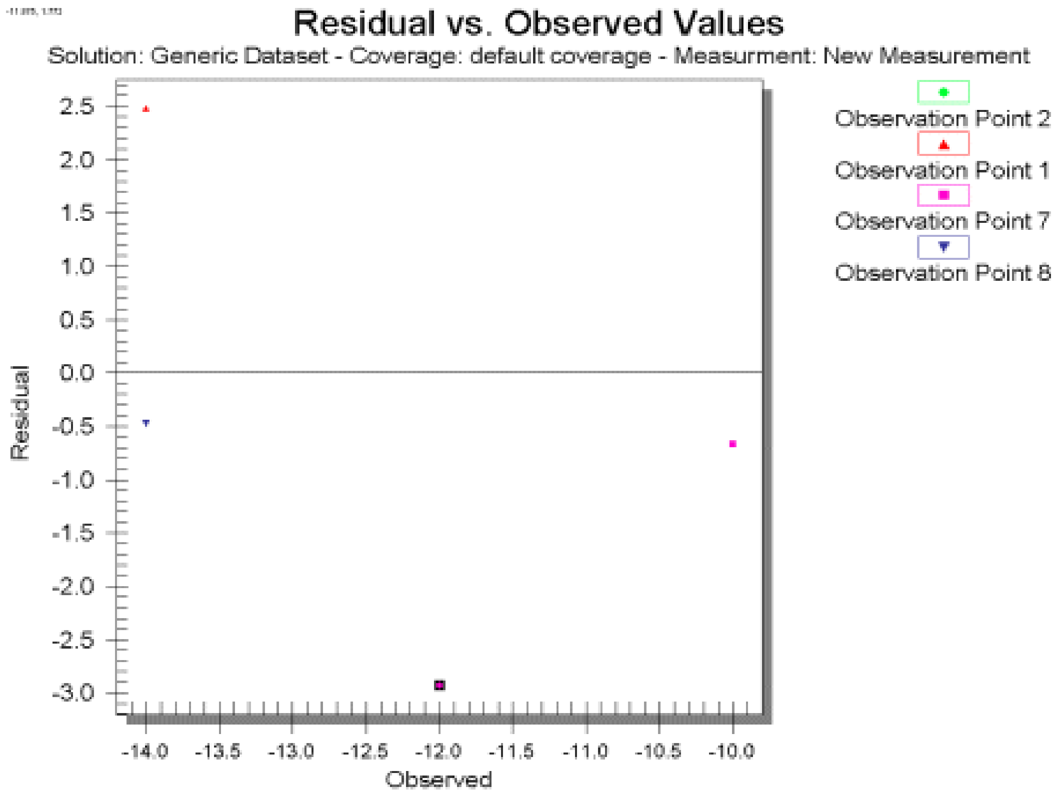
To open the Profile Customization Dialog, select Display | Plot Display Options or right click on the plot and select "Display Options".

Related Topics

- Plot Window

Residual vs. Observed Data

A Residual vs. Observed plot is used to display how well the entire set of observed values for observation points matches the solution data. On this plot is drawn a horizontal line along an error of zero, representing what would be a perfect correspondence between observed data and solution values. Then, one symbol is drawn for each observation point at the intersection of the observed and residual (computed-observed) values for the point. This plot can show the trend of the solution values with regards to matching the observed data. Only those points whose value is specified as observed for the selected data type will be shown in the plot. These plots are created in the Plot Wizard setting the Plot Type to Residual vs. Observed. A sample plot is shown in the figure below.



Residual vs. Observed Plots

After the plot type is set in Step 1 of the *Plot Wizard*, next to move to Step 2 where the plot options will be available:

Coverage – Displays the name of the coverage where the current data for the plot is coming from.

Measurement – This is the name of the current measurement, created in the *Feature Objects* | *Attributes* dialog, being plotted.

Feature Objects – Displays which feature object is utilized in the current plot, points or arcs.

Related Topics

- Plot Window

Time Series

The time series is similar to the XY series with a few differences.

1. Times series can support more than 2 columns of data.
2. The time series group can be specified (i.e. velocity, xy), when available.
3. When using times, the reference time can be specified.
4. The units of each column can be specified (from a list) and the column data will be converted when switching between units.

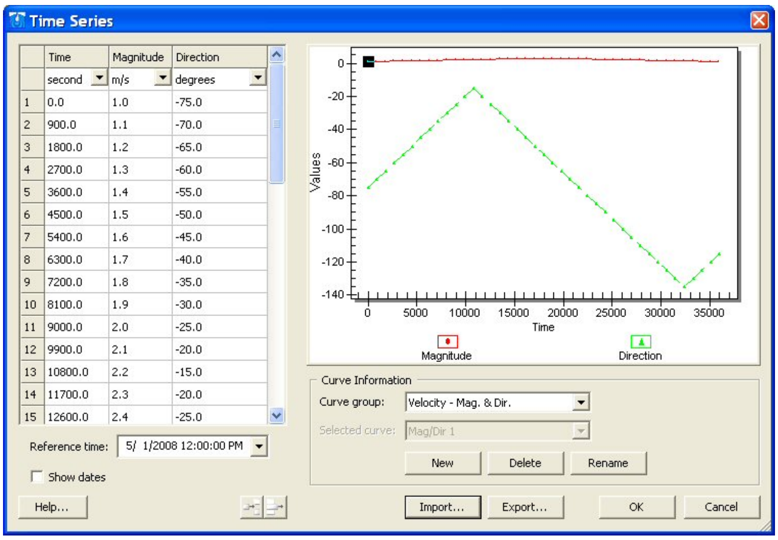
The time series editor can import xy series.

The time series assumes that all angles used are specified in the cartesian system.

For information about ADH model specific curve groups, see ADH Time Series.

Dialog Description

Curve Information



Time Series Dialog

Curve group field specifies the current group. This may be a combo box (depending on how the window is accessed) which allows the other groups to be selected.

Selected curve field specifies the current curve loaded. This may also be a combo box containing multiple curves. If this field is empty, then no curves exist in the current curve group.

New... button adds a new curve to the current curve group and opens a window in which the name of the new curve is specified. The name will appended if necessary to ensure uniqueness within the curve group. The new curve will be selected and appear in the *Selected curve* field.

Delete button removes the currently selected curve from the curve group. This button is only enabled if a selected curve exists.

Rename... button opens a window to specify a new name for the selected curve. This button is only enabled if a selected curve exists.

Plot type field specifies the current plot format. The available plot types are determined by the format of the current time series. The options for displaying the plots are as follows:

1. Scientific – This displays the data on a traditional XY plot.
2. Multi axes – This displays each column of the dataset on a separate Y axis.
3. Rose – This filters the data into vector data and then displays a rose plot of the binned data.

Various plot options, such as legend style and numerical precision, are accessed by right-clicking on the plot. It is important to note than when plotting data, the first column of the time series is always assumed to be the X values. This field is only enabled if a selected curve exists.

Curve Data

Spreadsheet lists the data of the selected curve. The column types are determined by the current curve group.

Attributes... button opens the selected curve's attribute window. This button is only enabled if the current curve group includes attributes. For information about ADH model specific curve group attributes, see ADH Time Series Attributes.

Insert New Row Above and ***Delete Row(s)*** toolbar assists in editing the spreadsheet rows. The tools are only available if there is a valid cell selection that excludes the title, units, and empty rows.

Reference time field specifies the date and time the selected curve begins out. This field is only visible if one of the curve groups available (listed in the ***Curve group*** field) allows time referencing and the field is enabled if the current curve group allows it.

Show dates check box specifies whether the time values of the curves are displayed in the spreadsheet as date and times instead of offsets from the reference time. This control is visible and enabled based on the same requirements as the **'Reference time**' field.

Miscellaneous (Outside of any group)

Import... and ***Export...*** buttons read and save, respectively, Time series (*.tsd) and XY series (*.xys) files.

Related Topics

- Compass Plot
- Spatial Data Coverage
- Coverages

Time Series Data File

The Time Series Data file format provides a means of transferring data to and from SMS. It is a simple ASCII file format that defines the type of data and its time reference.

The file follows the following format:

TIME_SERIES
Series Type Curve Name NCols NVals Reference Date

```
          Date 1          Value 1A         Value 1B

          Date 2          Value 2A         Value 2B

          Date 3          Value 3A         Value 3B

          Date 4          Value 4A         Value 4B

          .

          .

          .

          Date NVals      Value NVals      Value NVals
```

The following illustrates a sample file:

TIME_SERIES "Mag/Dir 1" "Velocity - Mag. & Dir." 3 41 "05/01/2008 12:00:00"

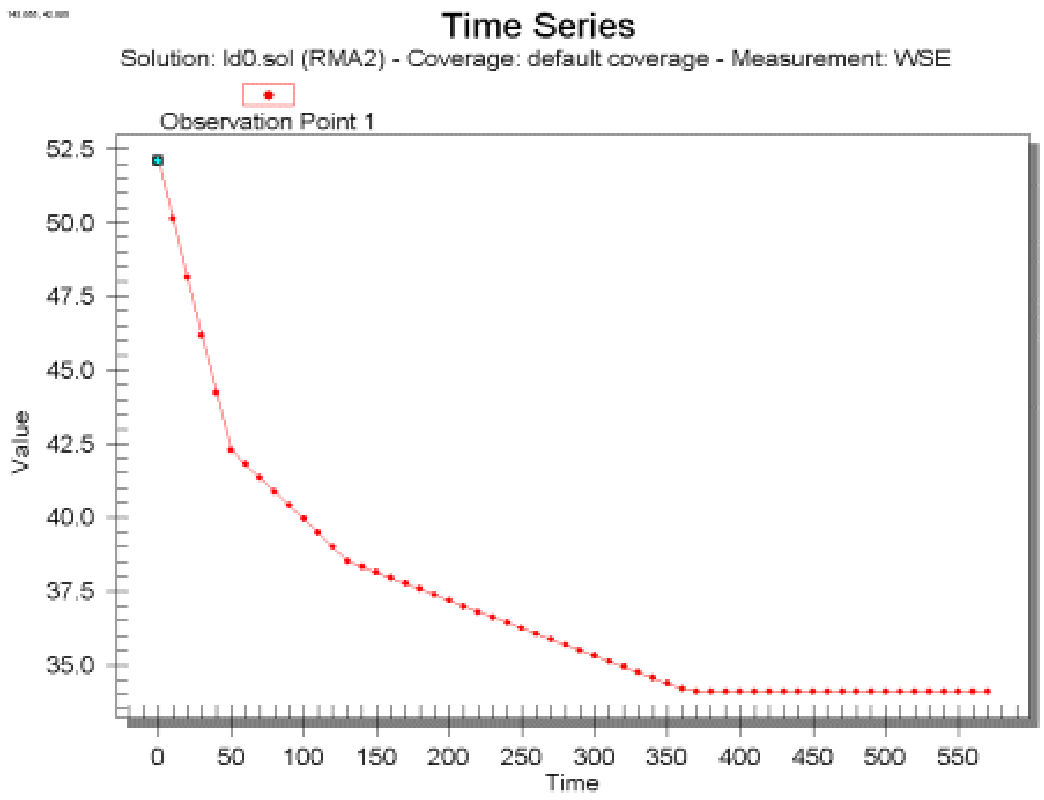
```
          0.0             1.0             -75.0
        900.0             1.1             -70.0
       1800.0             1.2             -65.0
       2700.0             1.3             -60.0
       3600.0             1.4             -55.0
       4500.0             1.5             -50.0
       5400.0             1.6             -45.0
       6300.0             1.7             -40.0
       7200.0             1.8             -35.0
       8100.0             1.9             -30.0
       9000.0             2.0             -25.0
       9900.0             2.1             -20.0
      10800.0             2.2             -15.0
      11700.0             2.3             -20.0
      12600.0             2.4             -25.0
```

Related Topics

- Time Series
- Spatial Data Coverage
- Coverages

Time Series Plot

A Time Series plot is used to display the time variation of one or more scalar datasets associated with a mesh or grid at observation points in an Observation Coverage. In addition, if transient calibration data has been defined, a band can be shown which represents a time variant Calibration Target. Only transient data sets may be used in these plots. Time Series plots are created by using the Plot Wizard, found in the Display menu, and selecting Time Series from the plot type list in Step 1 of the Plot Wizard. A sample plot, with calibration target band, is shown in the figure below.



Time Series Plot Options

After the plot type is set in Step 1 of the Plot Wizard, the user defines the time interval and the scalar dataset desired for the plot in Step 2. When this is completed, click Finish and the plot will be generated.

Use active dataset – This option causes the plot to display the values of the active dataset for each observation point being plotted. When the active dataset changes, the plot is recomputed and updated.

Use selected dataset – This option causes the plot to display the values of one or more specified datasets for each point being plotted. Changing the active dataset does not affect the plot. Select the dataset from the list box by putting a check in the dataset's check box.

Use calibration data – This allows the user to display the calibration curve defined for each point. If you do not have calibration data for the entity, leave the box unchecked and the calibration data will not be displayed.

For more information concerning how to edit the Time Settings, see Plot Window.

Related Topics

- Plot Window

View Values Dialog

The View Values dialog displays the values used to create the plot. The values can easily be copied from the dialog and pasted into a spreadsheet program or document using the following steps:

- Select the cells of interest from the spreadsheet
- Press Ctrl + C – cell contents are now in the clipboard
- Select the paste destination
- Press Ctrl + V

Related Topics

- Plot Window

ARR Mesh Quality Assessment Plot

The Angle Representation Region (ARR) plot is used to assess the overall quality of a triangular mesh such as those used by ADH, ADCIRC and other numerical engines. When the plot wizard is selected, this option appears if the mesh module is enabled. Clicking finish in the plot wizard results in an ARR plot for the current unstructured mesh, loaded in SMS.

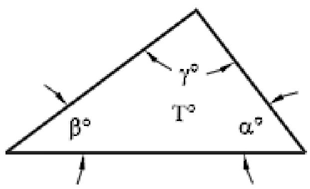
The plot includes the ARR region (defined below), a point for each element in the mesh, and three contour lines (0.3 in red, 0.45 in yellow and 0.6 in green) of the currently selected element quality measure (also defined below). As a general rule, elements with quality lower than 0.3 should be reviewed and improved (mesh editing) if possible.

The user can click on any point in the plot to see the element ID associated with that point and the six quality measure values for that element.

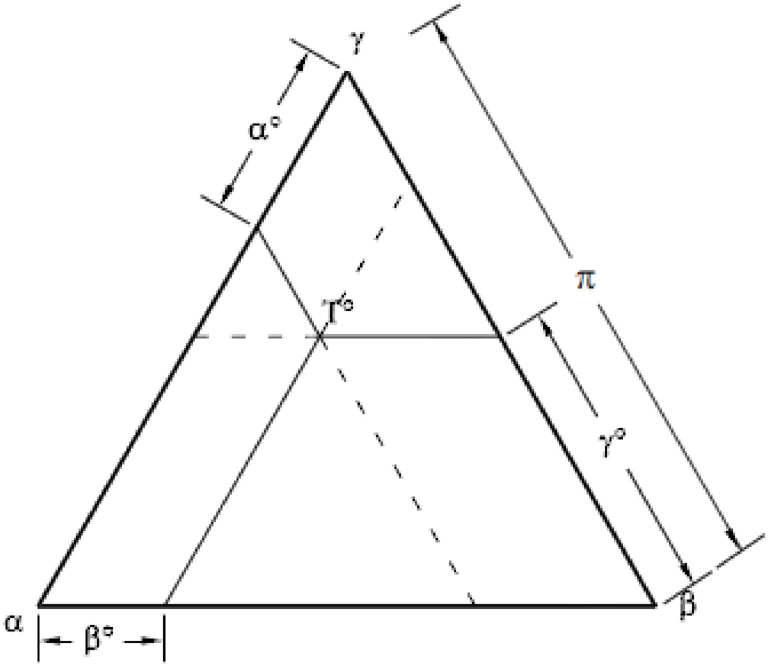
Once the mesh is edited in any way, the user can update the ARR plot by right clicking in the plot and selecting "Refresh". Until this is done, the plot will continue to reflect the mesh that existed when it was generated (or most recently refreshed).

This plot is based on a the publication in Communications in Numerical Methods in Engineerings, Volume 19 (2003) pp 551-561 by J. Sarrate, et. al. entitled "Numerical representation of the quality measures of triangles and triangular meshes". Several of the figures below are derived from this paper.

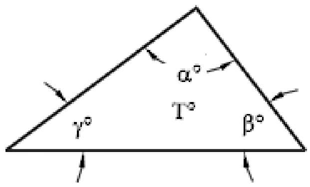
To assess the quality of a triangular mesh, such as those used by ADH or ADCIRC, the quality of each element is represented as a point, based on the interior angles of that element. These interior angles are labeled α, β, and γ as shown:



We can plot these three angles into an equilateral triangle



If we order the three angles so that α > β > γ as shown:



All of the points, will fall into the shaded portion of the equilateral triangle. This is referred to as the ARR region.

[File:ARRdomain.png]

The quality of the elements is further assessed by computing a quality measure from attributes of the triangle. These attributes include:

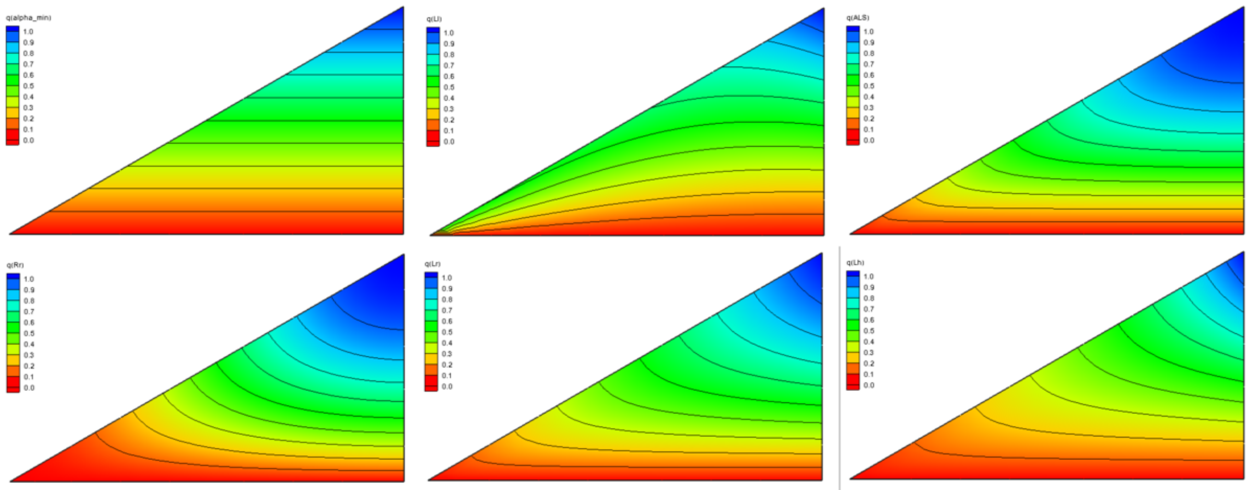
- The minimum interior angle α_{min}(γ from previous figure).
- The lengths of edges.
- The triangle area.
- The inner and outer radius.
- The minimum distance through the triangle (h_{min}).

These measures vary from 0.0 at the edges of the equilateral triangle to 1.0 at the center. The measures supported by SMS include:

$$q_{x_{min}} = \frac{3x_{min}}{\pi},\ q_{Ll} = \frac{l_{min}}{l_{max}},\ q_{ALS} = \frac{4\sqrt{3}A}{l_1^{\frac{2}{1}} + 1_2^{\frac{2}{2}} + 1_3^{\frac{2}{3}}}$$

$$q_{Rr} = \frac{2r}{R},\ q_{Lr} = \frac{2\sqrt{3}r}{l_{max}},\ q_{Lh} = \frac{2h_{min}}{\sqrt{3}l_{max}}$$

The following figures show how each of these quality measures cover the ARR



Related Topics

- Plot Window

2.2. Animation(Film Loop)

Animations

At a glance

Flow trace animation

- Several types of AVI animations (film loops) can be generated by SMS
- Transient data animation shows model changes through time (contours, vectors, etc)
- Flow trace uses vector data to generate flow paths through the geometry
- Drogue plots use user specified starting locations and show how the particles would flow through a vector field
- Multiple view animations show the data while transitioning between different views
- Plot window animations show plots changing through time

Animations in SMS provide a powerful tool for visualizing solution data.

Film Loop Setup Wizard

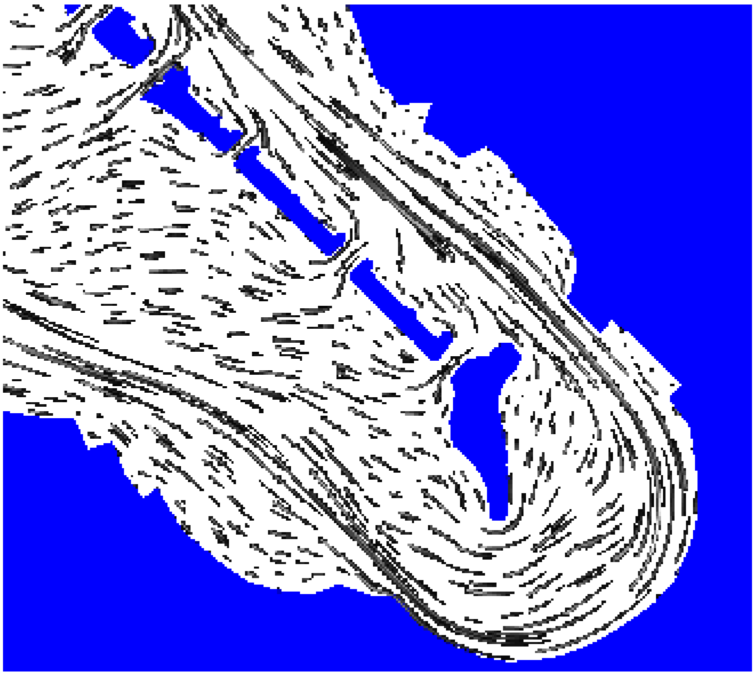
To create an animation select *Data* | *Film Loop* to open the Film Loop Setup wizard. The pages in the Film Loop Setup wizard include:

- General Options
- Display Options
- Time Step Options
- Multiple Views
- Drogue Plot Options
- Flow Trace Options

Animation Types

Flow Trace Animations

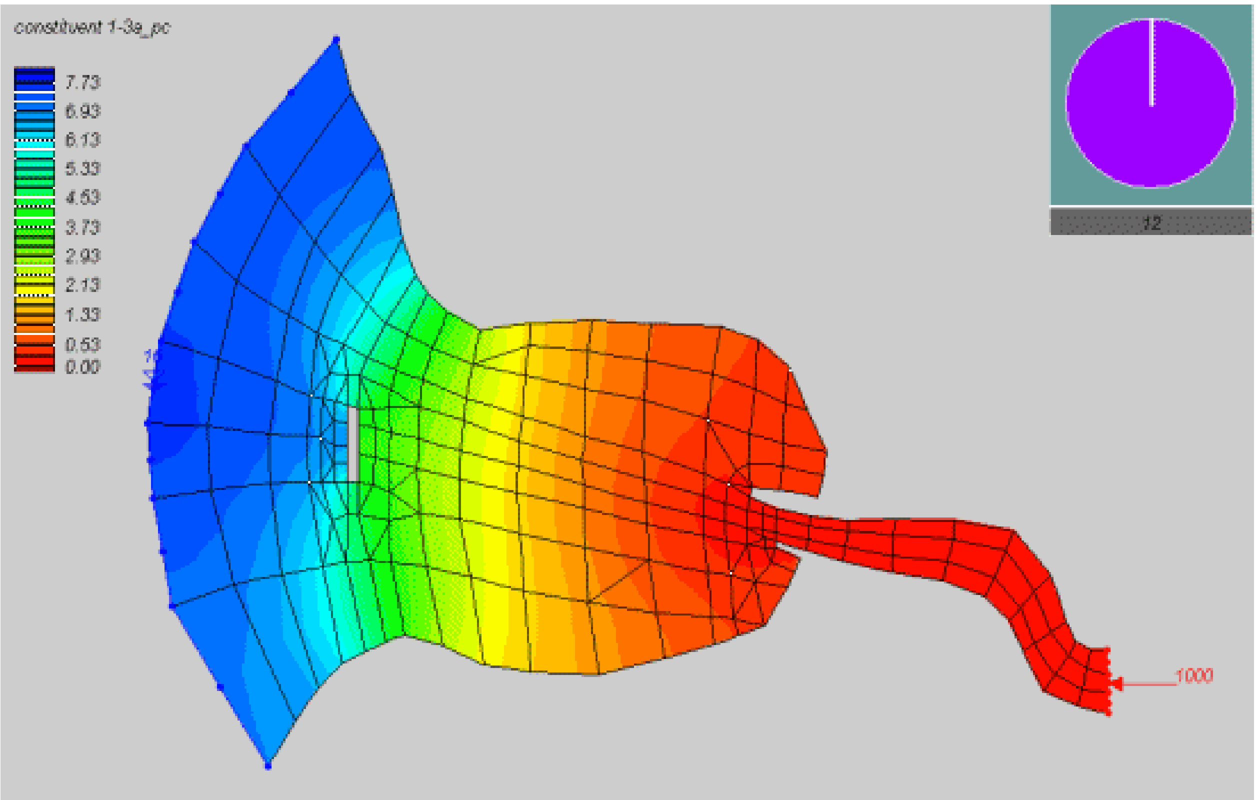
Flow trace animation is a technique used to visualize vector fields in SMS. It can be thought of as dropping tiny drops of dye into a fluid field in a random distribution and watching the flow pattern created. The process can also be thought of as creating particles of zero mass, and letting the vectors in the vector field be forces pushing the particles around. The *Flow Trace* portion of the *Film Loop Setup* dialog allows the user to control the flow trace. This entire portion of the dialog is disabled if no vector data exists for the current data set. The top radio group allows the user to specify whether the flow trace should be created for a steady state or dynamic system. Below this the user can specify the density of particles or dye droplets by specifying the average number of particles for each cell or element. The number of frames required for a droplet to become dispersed is represented as a portion of the animation in the Decay ratio field.



The path of each particle is defined by tracing the particle. A starting position is defined randomly in the mesh or grid. Successive particle locations are computed by applying the forces of the vector field to the current location. At the new point, the velocity and direction are sampled. If the particle has traveled farther than the Flow trace length limit, or the velocity has changed more than the Velocity difference limit, the step is broken into two steps of half the step size. This process is repeated, until a sequence of valid points within the limits are defined for each frame. Therefore, the smaller the values of the Flow trace length limit and Velocity difference limit, the more precisely the particles will imitate the vector field. Generally, the default values are sufficient.

The Average particle speed is used to scale the vector field, thus changing the distance each particle or droplet travels. This is useful for vector fields with extreme magnitudes. For a low magnitude data set, the particles may not move very far. While this sluggish motion is accurate for the data, scaling the vector field up, and exaggerating the motion causes the flow patterns to be more visible. Similarly, in high magnitude fields the particles may become long streaks and scaling the values down may result in a clearer picture of the flow patterns.

Transient Data Animations



Animation Clock

Since animations are simulating the passage of time, it is natural to display a clock, which indicates the time reference for each frame of the animation. The *Display Clock* toggle controls whether a clock will be displayed. The *Options* button brings up the Legend Options dialog with a control to specify a digital clock face or analog.

Animation Time Control

Animation can be applied to any object with a dynamic dataset. The user defines the beginning and ending time for the animation sequence and the time step between subsequent frames. As each frame is generated, data values corresponding to the current time are loaded into memory and the image is redrawn using the current display options. The display options may be modified while setting up an animation using the display options button in the *Display Options* portion of the dialog.

The strip in the center of the *Data Options* portion of the *Film Loop Setup* dialog displays the allowable time values for the current data function(s) and the selected range to be animated. The user can select a time range to animate graphically on this scale, or explicitly in the edit fields below the time step strip. The legal time range displayed in the strip is based on the current scalar and vector data set(s). SMS allows animation of only scalar or vector data while the other remains constant. This normally is only used when a static field such as elevation is displayed with a varying velocity field or a static velocity field is displayed over a changing scalar field such as constituent dispersion or sediment deposition.

The total number of frames generated in the film loop can be defined by either matching the time steps (one frame per time step) or by using a constant interval (e.g., one frame for every two-hour interval). If the *Match Time Steps* option is chosen, extra frames can be created between each time step using linear interpolation of the data values at the specified time steps.

Related Topics

- Visualization

Film Loop Display Options

This page of the *Film Loop Setup* wizard allows you to set up the film loop clock options. You can place the clock on any corner of the screen, set its size, and set the font that is used for the digital clock. In SMS version 9.0, you can set additional options for the clock position, progress bar and clock style.

This page also gives you access to the SMS *Display Options* property sheet. These options only affect the display of *Scalar/Vector Animations*.

AVI Codecs

Starting in SMS 10.0 you will be able to choose what Codec you use to create your AVI movie. SMS will search your computer for all compatible codecs and they will be available in the pull down menu. The choice of codec will determine both the quality and size of the resulting avi file.

FFDSHOW Video Codec

Ffdshow Video Codecs create a much sharper, smoother image than the SMS default codec (Microsoft Video 1), and is therefore more desirable to use when making film loops. ffdshow has an extensive list of codecs from which you can choose. We have had good success with the Divx codec. The H.263+ codec creates nice animations but they will not play within PAVIA (default for playing vidoes from SMS). See below for some ideas of alternate video players. The H.264 codec is popular but doesn't always work.

Installation

In order to use an ffdshow codec, you must first download ffdshow. If you installed the 32-bit version of SMS, ffdshow was installed unless you turned off the option during installation.

If you don't have it installed, to download ffdshow, go to http://ffdshow-tryout.sourceforge.net [1].

- Make sure that the program will be saved on your local disc (C:) program files.
- During installation, be sure to specify that you want the VFW interface.

Using FFDSHOW

There are two steps to using the ffdshow codecs for animations generated in SMS. First you have to use the ffdshow program to set the options for the video encoder. Secondly, you choose the ffdshow codec in the filmloop wizard from inside SMS. Detailed steps are given below:

- Once installed, click on the windows 'start' button and search for 'VFW Configuration'. Click on it.
- The ffdshow video encoder configuration dialog will appear.
- Set the Encoder box to which ever codec you would like to use. (Some codecs will not function because of incompatible requirements with the SMS filmloop generation code). Options of the corresponding Fourcc will appear in the Fourcc box. Pick one and Click the 'Apply' button. Click 'OK' to close the dialog.
- Once the Codec type is set, go to the Film Loop Setup - Display Options dialog in SMS (Data → Filmloop) and change the Codec type to 'ffdshow Video Codec.'

Note: Any codec that you use needs to be on the computer that will be playing the animations. If you want to move your avi's to a different computer, make sure that the computer you are moving them to has ffdshow. Otherwise, you will have to download ffdshow to play the animations.

Alternate Video Players

- KMPlayer [2]
- VLC [3]
- Windows Media Player [4] – Doesn't have much to control playback of videos so the others might be better options.

Related Topics

- Animations

References

[1] http://ffdshow-tryout.sourceforge.net/
[2] http://kmplayer.kde.org/
[3] http://www.vlcmediaplayer2011.com/
[4] http://windows.microsoft.com/en-US/windows/products/windows-media-player

Film Loop Drogue Plot Options

This page of the *Film Loop Setup* wizard allows you to set up color options that pertain only to drogue plots. A color ramp can be set up to display points in a color based on either its current velocity or the total distance it has traveled. The minimum value is always zero so the maximum value defines the range to be used for the specified color ramp. You may need to experiment with this maximum value before you get something that you like for a specific model. The head of each particle can be from one (1) to six (6) pixels in size. The maximum tail length is specified in hours and can fade to black or remain solid. If you don't want to have a tail, then set the fade time to zero (0.0).

You can specify the background to be either a solid color or an image that you have already opened and registered. If you don't have an image open, then that option is not available. The background of the model domain is always black.

The final option on this page allows a statistical report to be written while the particles are computed.

Related Topics

- Animations
- Color Ramp Options
- Particle Report

Film Loop Flow Trace Options

This page of the *Film Loop Setup* wizard allows you to set up options for the particles in the flow trace. The following options are available:

- **Particles per object** – Increasing this value increases the total number of random particles that get created and distributed throughout the domain. For a finite element mesh, the number specified is multiplied by the number of elements to determine the number of particles to be distributed over the domain.
- **Decay Ratio** – This defines how quickly the particle's tail decays and should be a value between zero (0.0) and one (1.0). A larger value produces particles with longer tails. A value of 1.0 indicates that it will take 100 percent of the film loop time for a end of a tail to fade away.
- **Average particle speed** – This provides a means of magnifying or reducing the activity in the domain. The particles will be traced through the domain with the velocity of the current vector functional dataset. The velocity is assumed as pixel space units. No time match between distance on the screen and velocity is attempted because the time between flow trace frames is not explicit. If an exact velocity in distance is desired, the user must compute the scale value and specify it here. Normally, for visualization purposes, experimentation with this parameter will generate the desired results. Another option would be to use the drogue animation tools.
- **Flow trace length limit** – This specifies a maximum distance the particle can travel in a single numerical integration step. When computing the numerical integration, if a particle travels more than this distance, the integration step is reduced to produce a more accurate particle path. Decreasing this number causes slower integration, but a more accurate path.
- **Velocity difference limit** – This specifies a maximum change in speed the particle can experience in a single numerical integration step. When computing the numerical integration, if a particle speed changes more than this limit, the integration step is reduced to produce a more accurate particle path. Decreasing this number causes slower integration, but a more accurate path.

SMS will use the current background as the background for the drogue plots. The model domain is always black and particles are always white. These two options cannot be changed.

Related Topics

- Animations

Film Loop General Options

Overview

The *Film Loop Setup* wizard is used to create the following types of video animation files:

- AVI, or audio video interleave [1] (*.avi)
- Google Earth© KMZ (*.kmz) – See Google Earth© KMZ file export requirements section below.

The wizard is invoked by choosing the Film Loop option from the Data menu. When the wizard is successfully completed, the animation is generated according to the specified options. The animations are then opened and displayed. AVI files are displayed in the Play AVI Application (PAVIA). Google Earth© KMZ files are displayed in Gooogle Earth©. the Play AVI Application is included with SMS. Google Earth© must be downloaded and installed separately. See http://earth.google.com/ [2] for information on obtaining Google Earth©.

The *General Options* page allows you to specify the following:

- File name for each video animation file type being exported
- Film loop type:
  - **Transient Data Animation** – To use this option, you must have opened a dynamic solution file. This will show how contours and/or vectors change with time by displaying a sequence of images, one for each time step.
  - **Flow Trace** – To use this option, you must have an available vector data set, such as velocity. This animation randomly distributes particles throughout the domain and shows their path through time.
  - **Drogue Plot** – To use this option, you must have an available vector data set, such as velocity, and you must have created points and/or arcs in a Particle/Drogue coverage with the Map module. This option is similar to the Flow Trace, except that particles are initially placed at feature points and at each vertex of feature arcs in the selected coverage.
  - **Multiple Views** – This option creates an animation of a single time step from one rotated view to another. A viewing path is created with any number of bearing/dip pairs. Multiple View film loops can not be exported to a Google Earth© KMZ file.
  - **Plot Window** – This option allows you to animate a plot window, such as how a functional value across an observation arc changes through time. Plot Window film loops can not be exported to a Google Earth© KMZ file.

Google Earth© KMZ file export requirements

The following requirements must be met to export Google Earth© KMZ file

- Must be in plan view
- Must use a Global Coordinate Projection (not local)
- Film loop types which can be exported to KMZ
  - Transient data animation
  - Flow trace
  - Drogue plot
- Film loop types which cannot be exported to KMZ
  - Multiple Views
  - Plot window

Play AVI Application (Pavia)

Controls exist within the application to play, stop, and step the animation. When an animation is stopped on a frame, the frame's image can be copied to the clipboard using the Copy button. The pavia.exe file is a separate application and can be copied to other computers to display existing animations on computers that do not have SMS installed. This is useful for displaying animations during a presentation or sending the animation to another individual or entity. You do not have to close the pavia application before creating another animation in SMS, but you will not be able to create an animation with the same file name as an opened animation.

Related Topics

- Animations

External Links

- Google Earth© Application [2]

References

[1] http://en.wikipedia.org/wiki/Audio_Video_Interleave
[2] http://earth.google.com/

Film Loop Multiple Views

This page of the *Film Loop Setup* wizard allows you to define the view path to be traversed for the animation. By default, the initial view is set to the window's current bearing and dip. You can add any number of views with any number of steps to the view. For your convenience, the bearing/dip pair can be displayed inside each frame of the animation.

Related Topics

- Animations

Film Loop Time Step Options

The *Time Options* dialog is used to specify the time range and time step to use in the animation. It is the second step of the *Film Loop Setup* wizard when exporting the following film loop types:

- Transient Data Animation
- Flow Trace – if using transient data
- Drogue Plot – if using transient data

The following time options are available:

- Film loop start time
- Film loop duration
- Number of Frames or Time Step Size

If the Time Settings are set to display as *Relative Time*, the zero time can be changed. If exporting to a KMZ file, the time zone can also be specified. Specifying the time zone is required if the model time is in a local time zone (as opposed to UTC) for Google Earth to display the correct times for the associated temporal data.

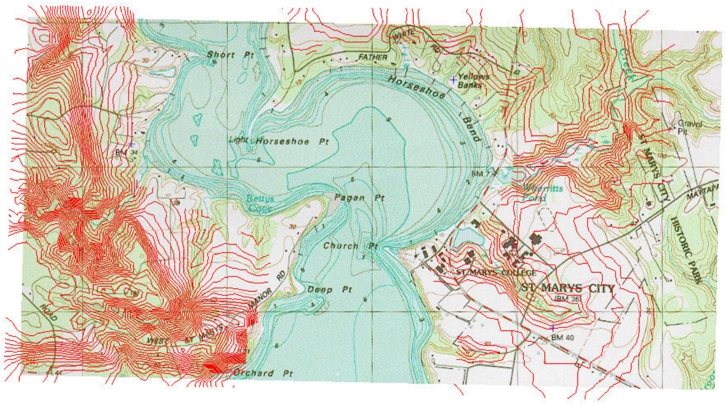
Related Topics

- Animations

2.3. CAD Data

CAD Data

CAD Support



Using CAD data in SMS

- AutoCAD DXF and DWG files can be read into SMS (support of DGN format is under development)
- Supports up to AutoCAD version 2007
- CAD data is displayed in 3D
- CAD data can be converted to map or scatter data

SMS can import CAD data from AutoCAD formats (DWG/DXF). CAD data in SMS can be converted and use in other modules. SMS data can be converted into CAD layers and saved in a supported format.

Importing

SMS can import DWG or DXF files via the File | Open command. If there is already CAD data in memory, SMS will replace the existing data with the data being imported. Currently, SMS cannot merge the incoming data with the data in memory.

Working with CAD data

The objects in a DWG or DXF file are organized into layers. The display of layers in a CAD drawing is controlled using the check boxes in the *Project Explorer*. Individual layers can be turned off/on or if you would like to turn off the display of all CAD data then uncheck the box next to the CAD folder.

Creating CAD data from SMS data

You can select either the DWG or DXF file types to save the CAD data. SMS objects must first be converted to CAD data before CAD data can be exported. To convert SMS data to CAD data, right click in the empty space at the bottom of the project explorer and choose *Convert To CAD*.

Delete Data

To Delete the CAD Data right-click on the CAD data folder in the tree and select *Delete* from the pop-up menu. If the CAD data was imported from a file, the file is not deleted from disk.

CAD → Map

CAD data can be converted to SMS feature objects by right-clicking on the CAD data folder in the *Project Explorer* and selecting *CAD → Map* command. CAD points are turned into points, CAD lines and polylines are turned into arcs, and CAD polygons are turned into polygons. The feature objects are added to the active coverage. Once converted, the feature objects can be used to build conceptual models.

CAD → 2D Scatter

A set of CAD 3D faces which have been imported to SMS can be converted to a 2D Scatter Set by right-clicking on the CAD data folder in the *Project Explorer* and selecting the *CAD → 2D Scatter* command.

Exporting

SMS data can be exported to a DWG or DXF file that can then be read into a CAD package. If there is CAD data in memory when a SMS project is saved, SMS creates a new DWG file from the CAD data. The file is put in the same folder with the other project files and named using the project prefix.

Alternatively, CAD data in memory can also be saved using the *Save As* command in the *File* menu.

CAD Data Right Click Menus

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a CAD Data item.

CAD Data Root Folder Right Click Menus

Right clicking on the *Cartesian Grid* module root folder in the Project Explorer invokes an options menu with the following options:

- Display Options

CAD Data Item Right Click Menus

Right clicking on a *Cartesian Grid* item in the Project Explorer invokes an options menu with the following module specific options:

- **Convert**
  - CAD → Map – Converts CAD data to Map Module data
  - CAD Faces → 2D Scatter Triangles – Converts CAD Faces to Scatter Module triangles
  - CAD Points → 2D Scatter – Converts CAD Points to Scatter Module vertices

Related Topics

- Project Explorer Right Click Menus
- Menu Bar

2.4. Coordinate Systems

Coordinate Systems

XMS programs work in a single coordinate system. The user selects what system will be used using the *Edit | Coordinate System* menu command.

Both a horizontal and vertical system are specified. Many numerical models work in a local system, so there may not be a problem if the user does not know what coordinate system the data is referenced to. If the desired model requires a global system, or the base data is referenced to more than one system, the modeler must determine the coordinate systems involved.

Since data can be gathered and referenced to various coordinate systems, XMS allows the user to convert data from one coordinate system into another. The Coordinate Conversions page describes how to do this. The available systems and units include:

Horizontal System

The options to the right of the dialog change as the horizontal system is changed. For example, the hemisphere is required for a Geographic system and a UTM zone is required for a UTM system.

- Local
- Geographic
- UTM
- State Plane
- CPP

Local is the default horizontal and vertical system. Unless the user desires to convert data to another system, it is not necessary to change the system from *Local*.

Horizontal Units

The horizontal units can be specified for all systems except Geographic, which uses decimal degrees. **The units set here are the units used by the models such as RMA2**.

- U.S. Survey Feet
- International Feet
- Meters

Ellipsoid

The Ellipsoid options are only available for non-NAD/HPGN/CPP systems. The ellipsoid can be selected for a region of the world. Changing the ellipsoid changes the minor and major radii of the earth, measurements used in performing conversions. To set the radii for an ellipsoid not included in XMS, select the *User Defined* ellipsoid and a button appears titled *Define*. Clicking on the button brings up a dialog where the radii are defined. The Major and Minor Radii (a and b, respectively) are defined in meters only. The Minor Radius can be input directly or defined by specifying the ellipsoid flattening (1/f) or eccentricity squared (e2) variables, where

$$\frac{1}{f} = \frac{(a-b)}{a} \quad \text{and} \quad e^2 = 1 - \frac{b^2}{a^2}$$

Vertical System

The supported vertical options are for North America. If a non-NAD/HPGN horizontal system is selected, this option is disabled.

- Local
- NGVD 29
- NAVD 88

Vertical Units

The vertical unit is only used for conversions. Area and volume calculations performed by XMS use the horizontal units.

- U.S. Survey Feet
- International Feet
- Meters

External Links

- Introduction to Tidal and Geodetic Vertical Datums and Datum Computations [1] – Presentations available here [2]

References

[1] http://www.ngs.noaa.gov/corbin/class_description/Tidal_Geodetic_Datums.shtml
[2] ftp://ftp.ngs.noaa.gov/pub/corbin/datum_training/

Coordinate Conversions

Converting data from one coordinate system to another can be done using the ***Edit|Coordinate Conversions*** menu command. The command may also be accessed by right-clicking on a single entity (grid, mesh, scattered data set ...) in the Project Explorer. This allows the user to convert just that entity from one system into another (typically to the current project coordinate system).

All data will be converted from the system on the left of the dialog to the system on the right.

"Convert From" System

The "Convert From" system defines the coordinate system the data is currently referenced to. When working from the *Edit* menu, this is the system XMS is working in and is dimmed by default because it is assumed you have already specified this system. When working from the Project Explorer, this is the coordinate system of the selected entity and must be selected.

"Convert To" System

The "Convert To" system defines the system you will be working in after the conversion. When the dialog is invoked from the *Edit* menu, this is selected by the user and all data is converted from the current system to this new system. When the dialog is invoked from the Project Explorer, the "Convert To" system is dimmed because this is assumed to be the system XMS is working in and all other data is already in this system.

Restrictions

Some conversions are not allowed, such as converting between a NAD and non-NAD system. A warning is issued when conversions are not allowed.

CPP Coordinate System

A CPP (Carte Parallelo-Grammatique Projection) system is a local system. The origin of the system must be defined in latitude/longitude decimal degrees.

The conversion from of a point from latitude/longitude to CPP is:

$$\mathrm{newpoint}_x = R * \left(\mathrm{point}_{\mathbf{longitude}} - \mathrm{origin}_{\mathbf{longitude}}\right) * \cos\left(\mathrm{origin}_{\mathbf{latitude}}\right)$$

$$\mathrm{newpoint}_y = \mathrm{point}_{\mathbf{longitude}} * R$$

The conversion of a point from CPP to latitude/longitude is:

$$\mathrm{newpoint}_{\mathbf{longitude}} = \frac{\mathrm{origin}_{\mathbf{longitude}} + \mathrm{point}_x}{R * \cos\left(\mathrm{origin}_{\mathbf{latitude}}\right)}$$

$$\mathrm{newpoint}_{\mathbf{latitude}} = \frac{\mathrm{point}_y}{R}$$

R = 6378206.4 m. (Clarke 1866 major spheroid radius)

Geographic Coordinate System

A Geographic system is a latitude/longitude system defined in decimal degrees. Supported Geographic systems include:

- NAD (North American Datum) 1927 and NAD 1988
- 33 world ellipsoids and a user defined ellipsoid (i.e., Clarke 1866, WGS 1984, etc.)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres).

Local Coordinate System

A local coordinate system is a system defined for a survey. Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). If a portion of the data for a project is referenced to a global coordinate system, and a portion is in a local system, a transformation to convert all the data to a single system must be defined. This can be done by defining the location and orientation of the origin of the local system with reference to the global system. This allows the data to be referenced back to the global system or to another local system.

Three scenarios are possible when dealing with local coordinate systems:

1. Global to Local – When converting from a global to a local coordinate sytem, the global coordinates of the origin of the local coordinate system must be defined using the Local Origin button in the lower portion of the dialog. This dialog can also be used to enter the angle of rotation of the local coordinate axes relative to the global coordinate axes. The angle is measured ccw from the positive x-axis.
2. Local to Local – When converting from one local coordinate system to another local coordinate system, the same approach is used as when converting from a global to local coordinate sytem. The Local Origin dialog is used to define the coordinates of the origin of the new coordinate system relative to the old coordinate system.
3. Local to Global – When converting from a local to a global coordinate sytem, the Local Origin dialog is used to coordinates of the local coordinate system (which is the old system in this case) relative to the new global coordinate sytem.

Single Point Conversion

The **Single Point Conversion** dialog allows a user to convert a coordinate (xyz point) between to coordinate systems. This dialog can be accessed from two locations within XMS programs:

1. *Edit* menu, *Convert Single Coordinate* item
2. Register Image Dialog

The dialog consists of two sides. On the left side, the original (or Convert from) coordinate system is entered. On the right side, the final (or Convert to) coordinate system is entered. The options for the two sides are described on the Coordinate Systems page.

The other items in the dialog include:

- **Enter coordinates** – Enter the original (or from) coordinates.
- **New Coordinates** – View the final (or to) coordinates.
- **Convert** – Perform the conversion.
- **Create Mesh Node/Feature Point** – Create a mesh node (if in the mesh module) or a feature point (if in the map module) at the final coordinates when OK is pushed.

Related Topics

- Coordinate Systems
- Coordinate Conversions

2.4.a. Projections

Projections

"Projection" refers to a map projection like UTM [1]. In XMS software, a projection can be associated with a project, and data can be reprojected from one projection to another. XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

Previous XMS software versions referred to projections as "coordinate systems" and reprojection as "coordinate conversion".

Project Projection

The current projection, or the projection currently associated with the project, can be specified via the *Edit|Projection* menu command. Changing the projection does not alter the XYZ coordinates of the project data.

Local Projection

Many numerical models work in local systems, and don't care how that system relates back to global coordinate systems (UTM, State Plane etc.). XMS software allows for local projections that are unrelated to any standard projection.

Reproject

Reprojecting means to convert data from one coordinate system to another. For example, a 2D mesh representing the ground surface may have XYZ coordinates in a UTM system and they need to be converted to a State Plane system to be consistent with other data. Reprojecting usually results in the XYZ coordinates of the data changing, although conceptually the data is in the same place with respect to the Earth, just in a different coordinate system.

There are three basic reprojection tasks that you can do:

- Reprojecting the entire project from one system to another
- Reprojecting one object from one coordinate system to the project coordinate system
- Single point reprojection, which allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection.

Either operation brings up a dialog with two projections specified. On the left, the "Convert From" projection defines the projection the data is currently in. On the right, the "Convert To" projection defines the projection the data will be in after the operation.

Reproject everything

Reprojecting everything can be done by selecting the ***Edit|Reproject*** menu command. This will convert all the data loaded into the XMS application from one projection to another. In this operation, the "Convert From" projection (left side of the dialog) is dimmed by default. The user selects a new projection on the right side that all data in the application will be converted to. This operation changes the "current" projection that is being used by XMS to the newly specified "Convert To" projection.

Reproject object

This command is done on a specific geometric object (grid, mesh, scatter set, ...) by right-clicking on the entity in the Project Explorer. The object is reprojected from some projection into the current projection being used by the XMS application. In this operation, the "Convert To" projection (right side of the dialog) is dimmed by default because it represents the current projection used by the project. It is assumed that the desire is to reproject data from another projection into the current projection. The current projection is not changed in this operation.

Single Point Reprojection

Single Point Reprojection allows you to enter the XYZ coordinates for a point in one projection and see what the new coordinates would be if the point was reprojected to a different projection. It also lets you create a feature point at the new location. This operation is accessed via the *Edit|Single Point Reprojection* menu command. It's also available in the Register Image dialog.

Restrictions

Some reprojections are not allowed, such as reprojecting between a NAD and non-NAD system. A warning is issued when the reprojection is not allowed.

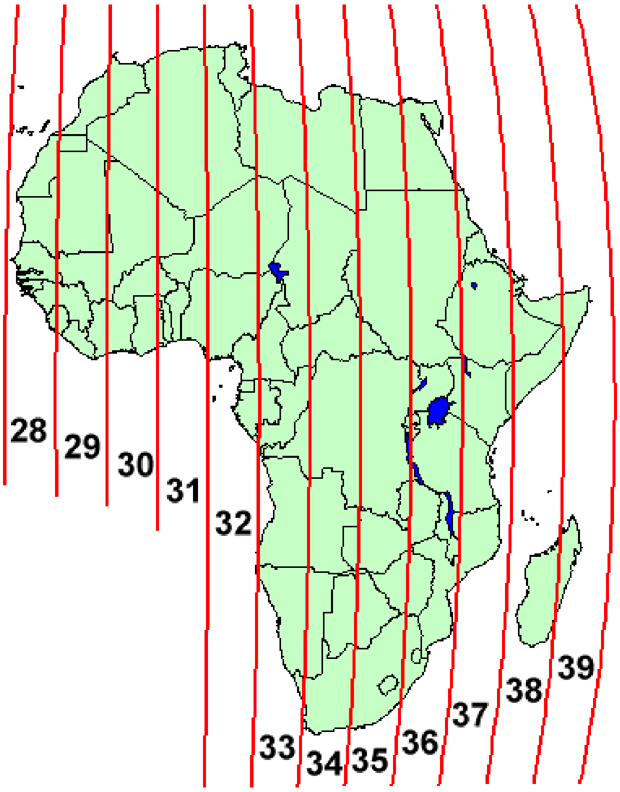
Supported Projections

XMS software utilizes the Global Mapper (TM) [2] library which supports hundreds of standard projections.

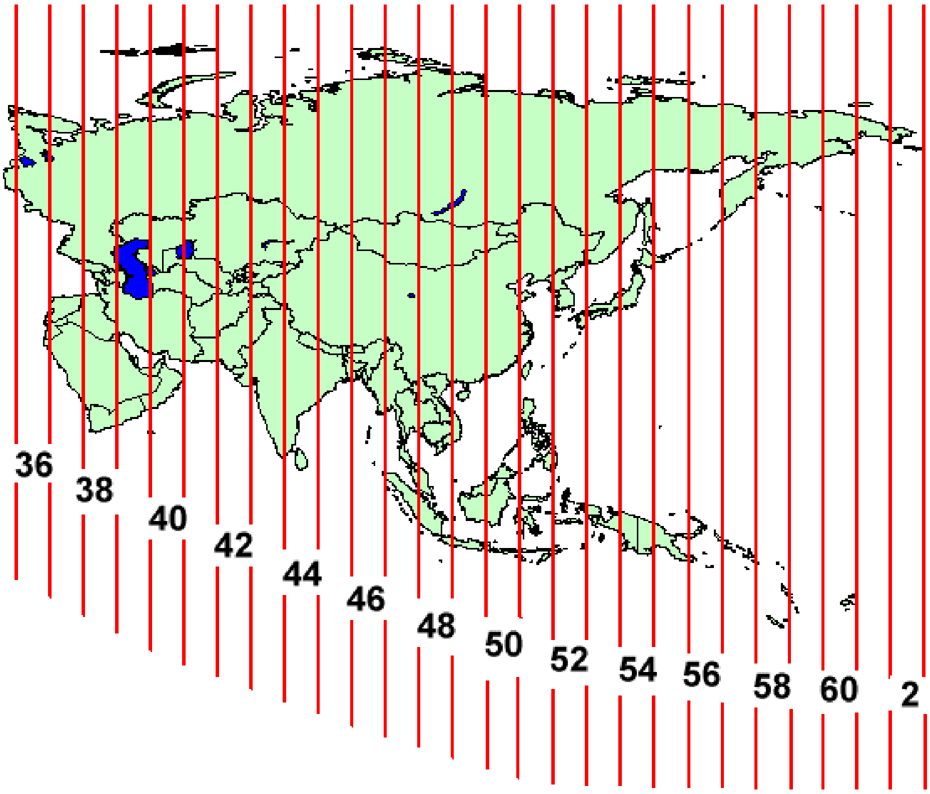
References

[1] http://en.wikipedia.org/wiki/Universal_Transverse_Mercator_coordinate_system
[2] http://www.globalmapper.com/

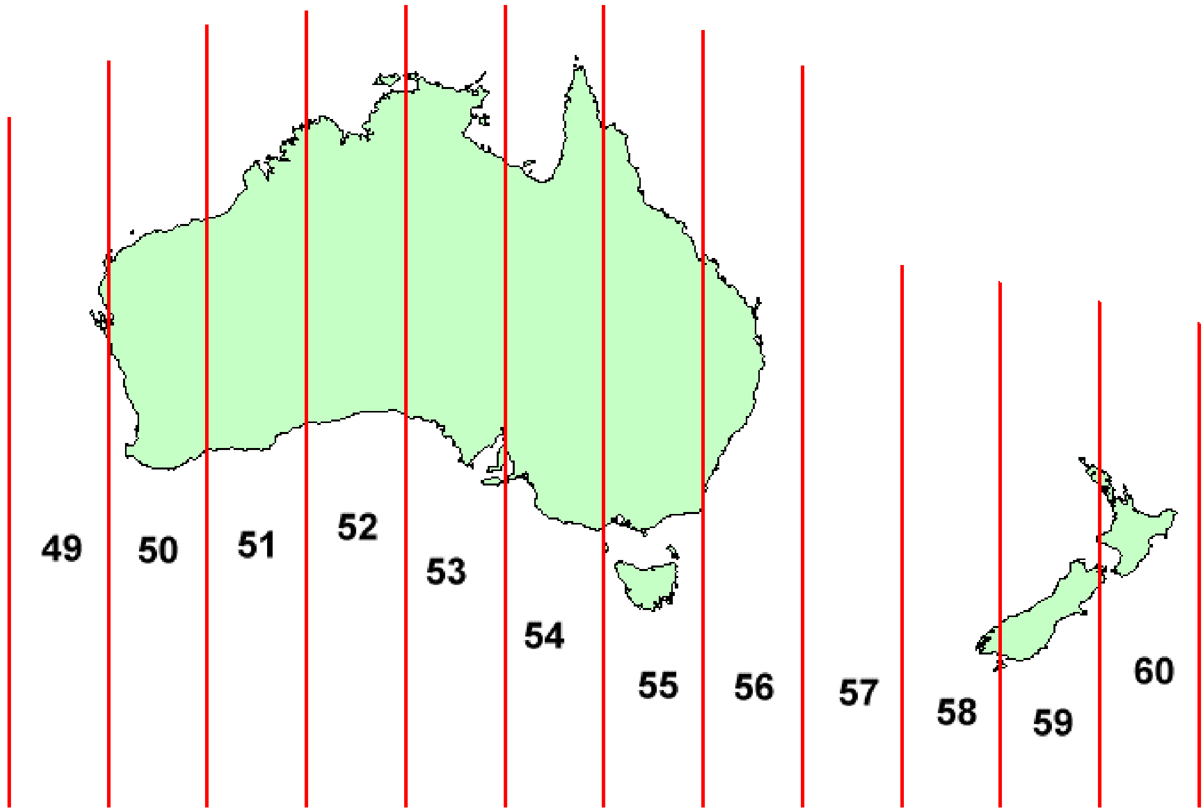
UTM Africa



UTM Asia



UTM Australia



UTM Coordinate System

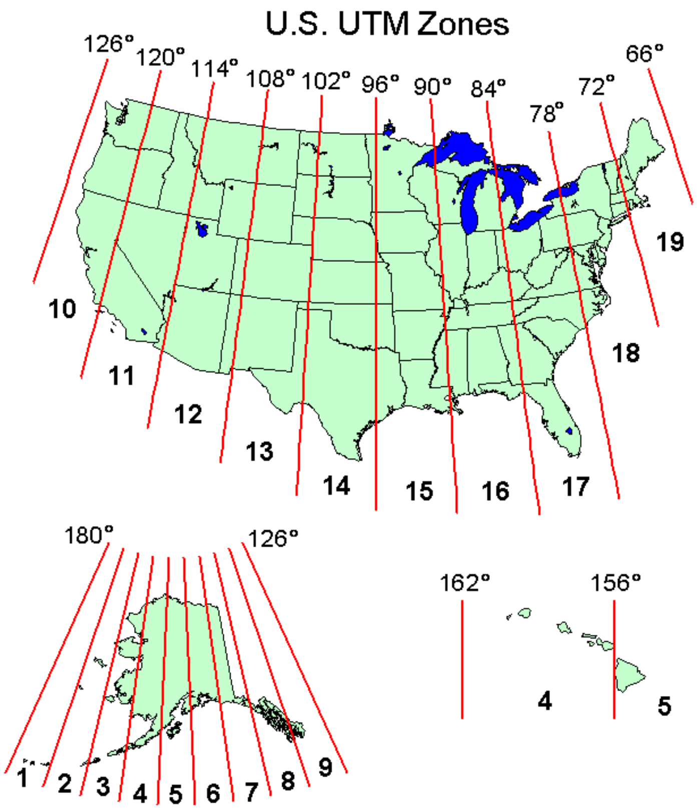
A UTM (Universal Transverse Mercator) system is a world-wide system defined in meters. The world is divided into 60 zones, 6 degrees of longitude, running from 84°N to 80°S latitude. Supported UTM systems include:

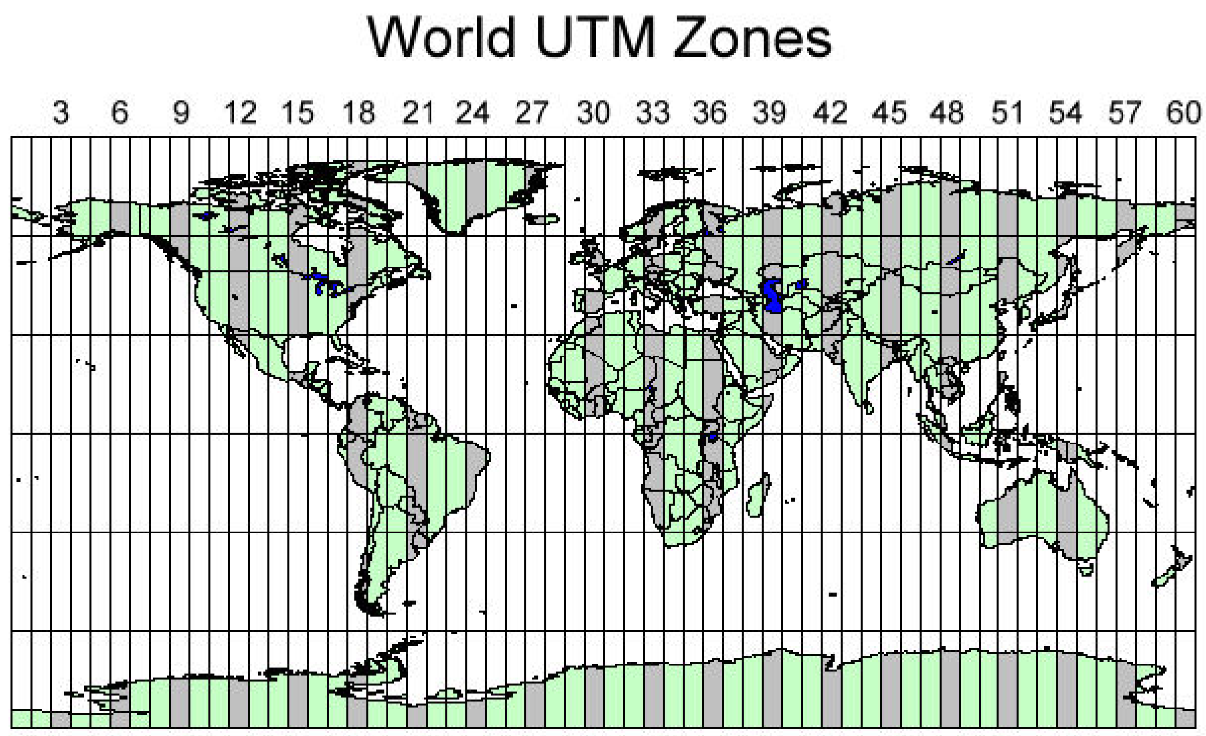
- NAD (North American Datum) 1927 and NAD 1988
- HPGN (High Precision Geodetic Network, now known as HARN - High Accuracy Precision Network)

The hemispheres are defined for non-NAD systems. The hemisphere cannot be changed for NAD systems (Northern, Western hemispheres). An additional HPGN zone must be defined for HPGN systems.

UTM zones for the southern hemisphere have a "false northing" of 10,000,000 meters at the equator with northings decreasing as you move south. This ensures all northings are positive in the southern hemisphere.

The US and World UTM Zones are shown below.





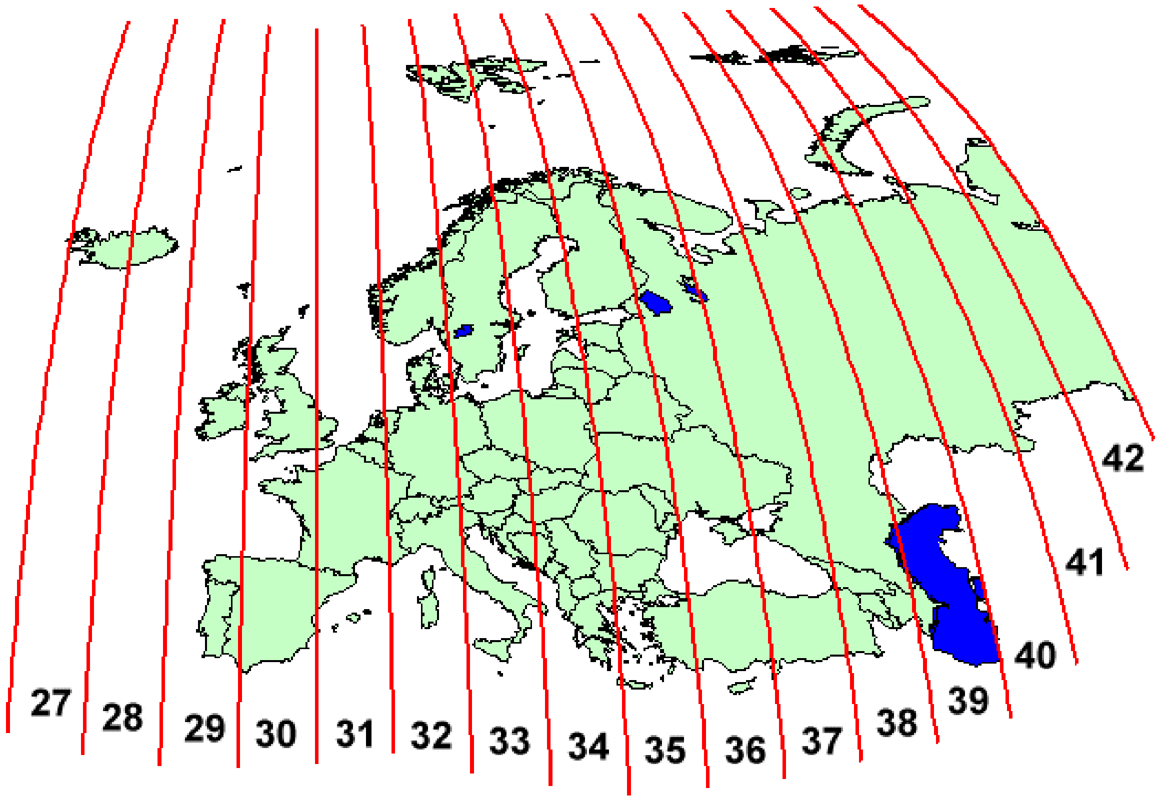
UTM Zones By Continent

- North America
- South America
- Africa
- Asia
- Europe
- Australia

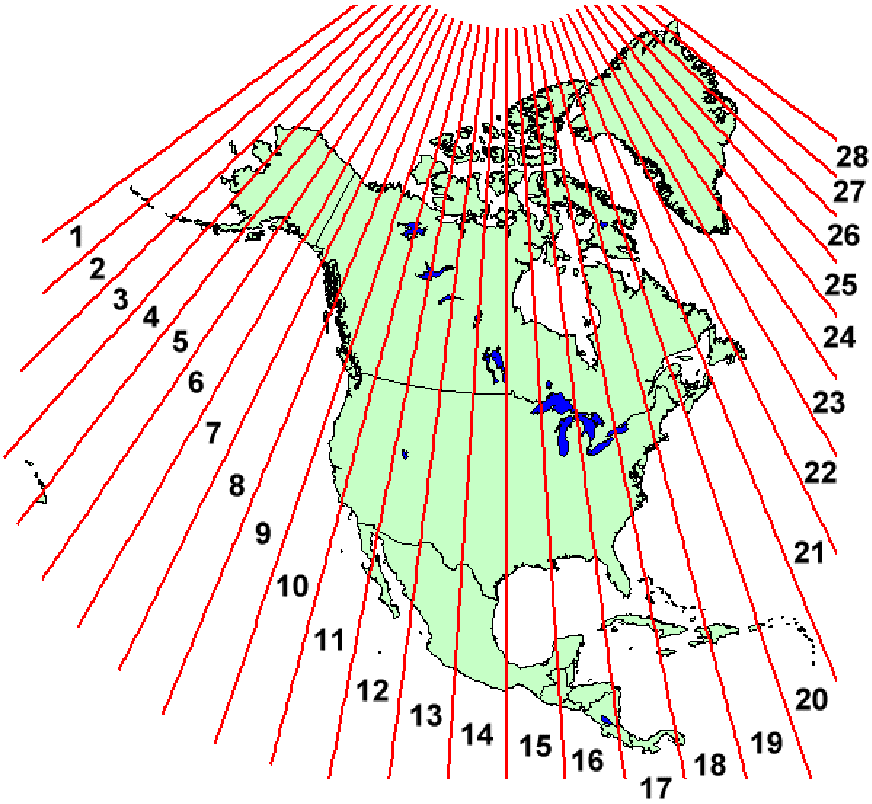
Related Topics

- Coordinate Systems
- UTM Coordinates [1]

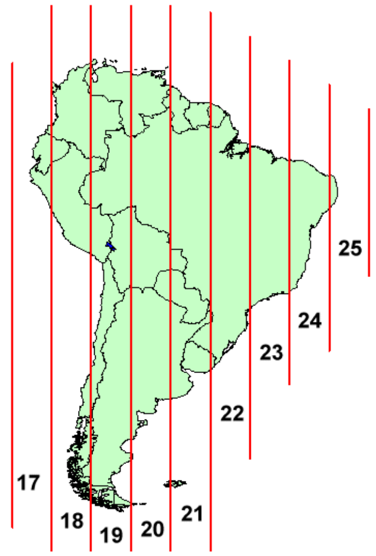
UTM Europe



UTM North America



UTM South American



2.4.b. State Plane Coordinate System

State Plane Coordinate System

A State Plane system is a coordinate system used in the US. Each US state is divided into one or more zones, known as State Plane zones. Supported State Plane systems include:

- NAD (North American Datum) 1927
- NAD 1983

Additionally, an HPGN (High Precision Geodetic Network, now known as HARN - High Accuracy Precision Network) zone can be specified for each state plane zone.

The boundary of most of the state plane zones remained the same from 1927 to 1983. The US State Plane Zones are shown in the map below. The boundaries are shown for each state plane zone by clicking on a region on the map. The boundaries that changed between 1927 and 1983 are highlighted for each state plane zone that changed.

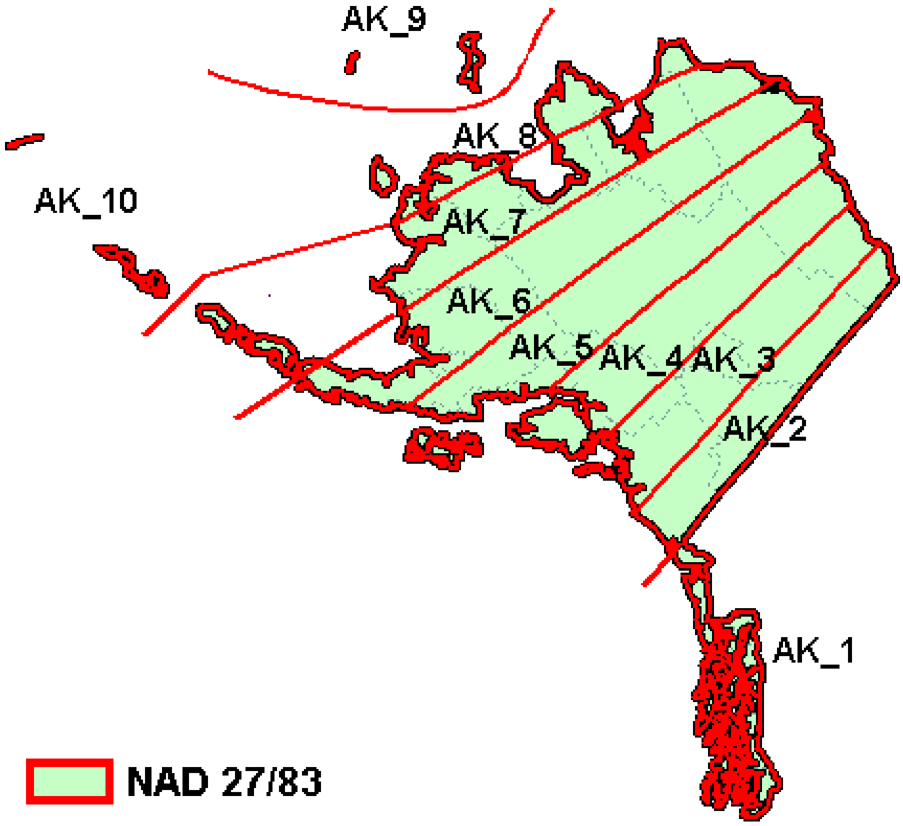
State Zone Maps

- Alaska
- Hawaii
- Mideast
- Midwest
- New England
- Northwest
- South Central
- South East
- Southwest
- Virgina Area

Related Topics

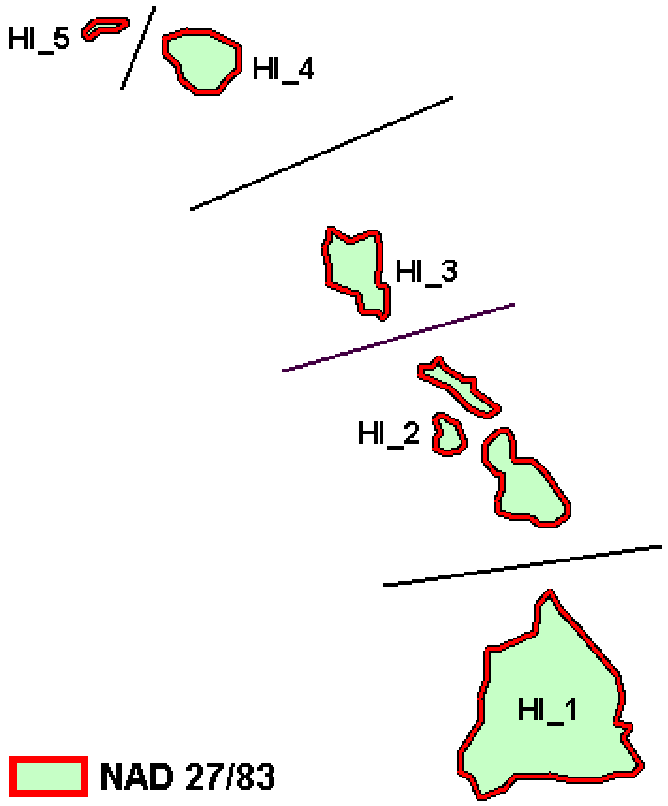
Coordinate Systems

Alaska State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID
Alaska 1	AK_1	5001
Alaska 2	AK_2	5002
Alaska 3	AK_3	5003
Alaska 4	AK_4	5004
Alaska 5	AK_5	5005
Alaska 6	AK_6	5006
Alaska 7	AK_7	5007
Alaska 8	AK_8	5008
Alaska 9	AK_9	5009
Alaska 10	AK_10	5010

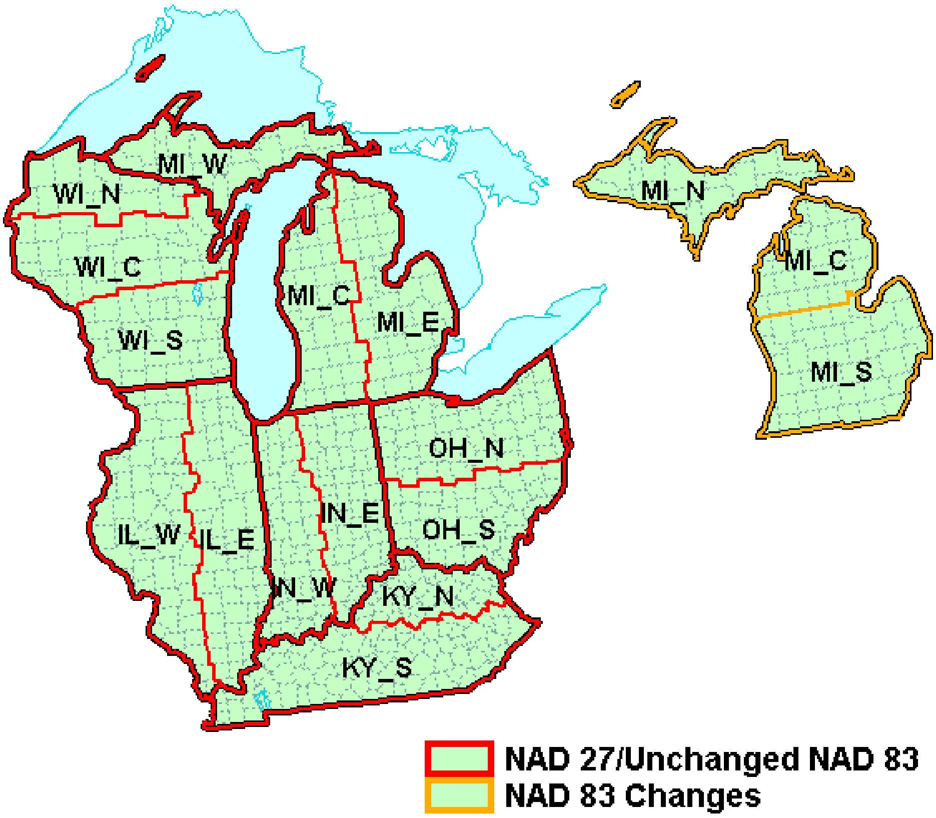
Hawaii State Plane



NAD 27 / 83

Zone Name	Map Code	Zone ID
Hawaii 1	HI_1	5101
Hawaii 2	HI_2	5102
Hawaii 3	HI_3	5103
Hawaii 4	HI_4	5104
Hawaii 5	HI_5	5105

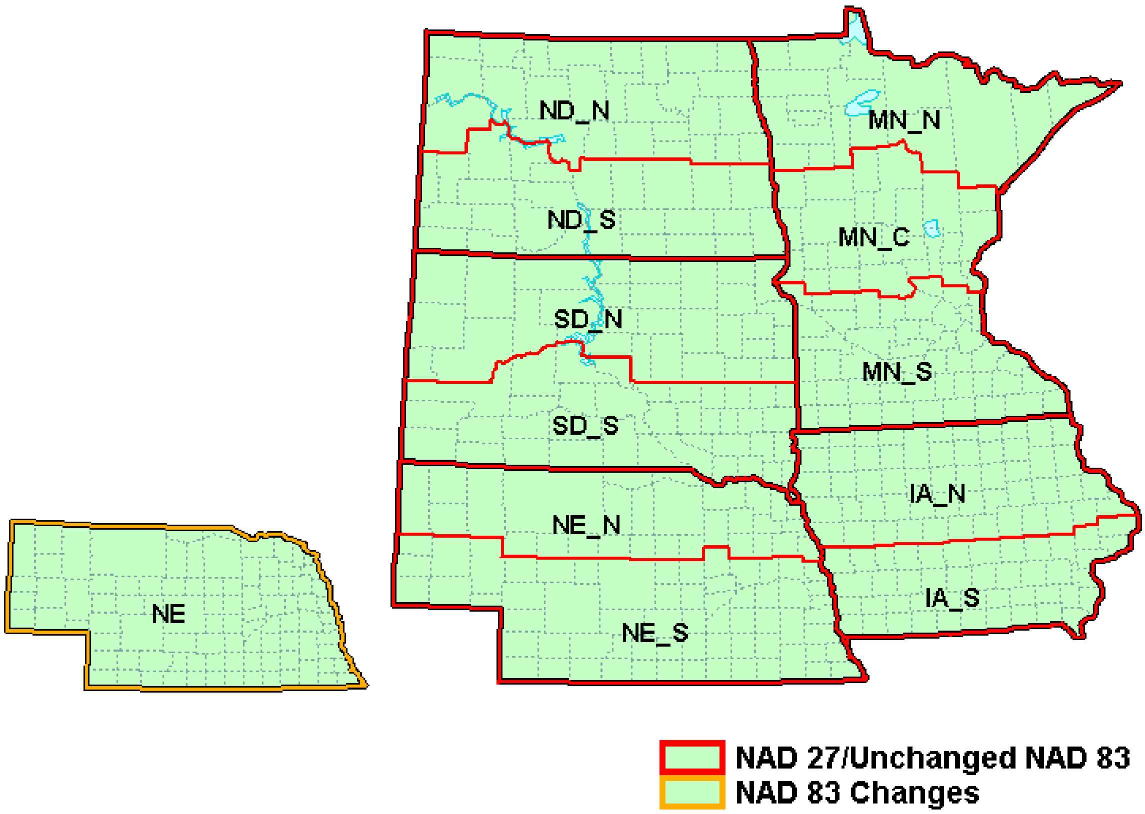
Mideast State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID

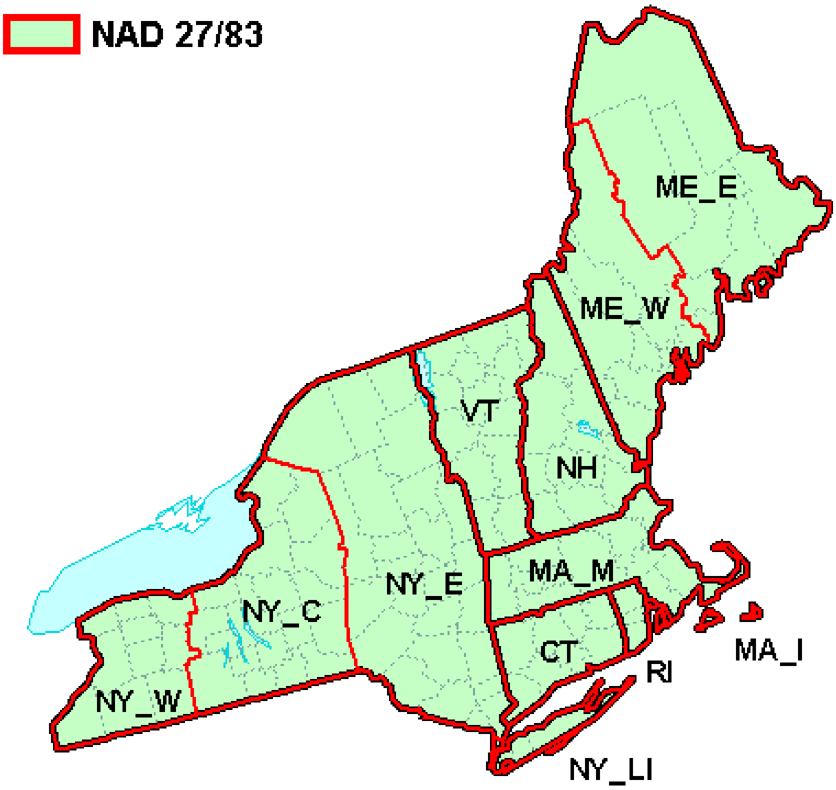
Illinois East	IL_E	1201
Illinois West	IL_W	1202
Indiana East	IN_E	1301
Indiana West	IN_W	1302
Kentucky North	KY_N	1601
Kentucky South	KY_S	1602
Michigan East (NAD 27)	MI_E	2101
Michigan Central (NAD 27)	MI_C	2102
Michigan West (NAD 27)	MI_W	2103
Ohio North	OH_N	3401
Ohio South	OH_S	3402
Wisconsin North	WI_N	4801
Wisconsin Central	WI_C	4802
Wisconsin South	WI_S	4803
NAD 83 Zone Changes		
Michigan North	MI_N	2111
Michigan Central	MI_C	2112
Michigan South	MI_S	2113

Midwest State Plane



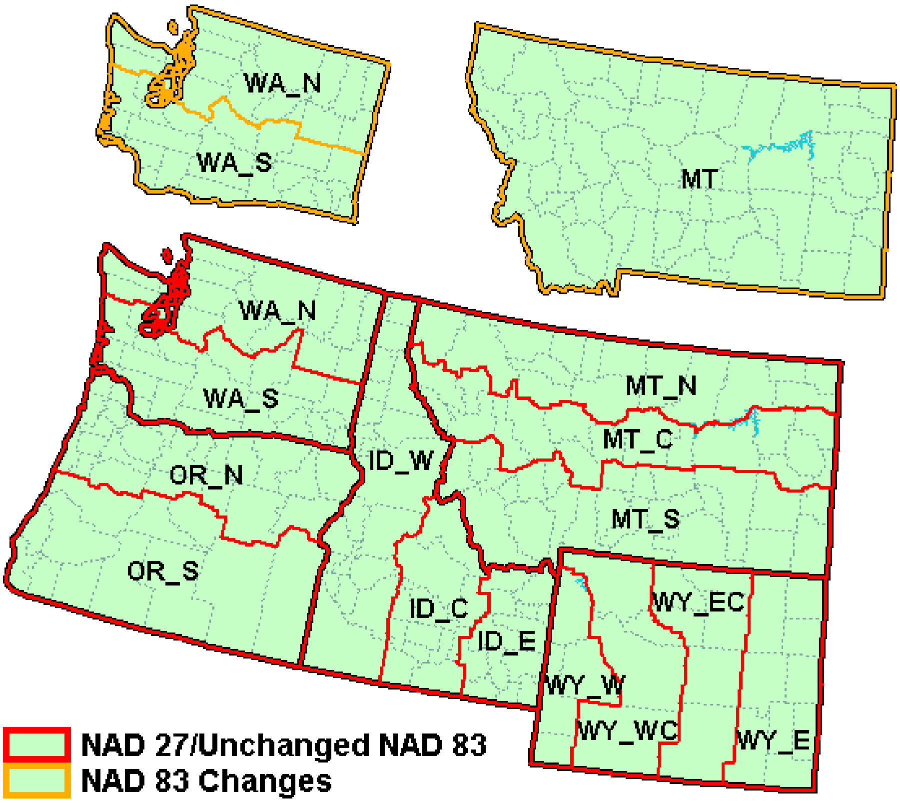
NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Iowa North	IA_N	1401
Iowa South	IA_S	1402
Minnesota North	MN_N	2201
Minnesota Central	MN_C	2202
Minnesota South	MN_S	2203
Nebraska North	NE_N	2601
Nebraska South	NE_S	2602
North Dakota North	ND_N	3301
North Dakota South	ND_S	3302
South Dakota North	SD_N	4001
South Dakota South	SD_S	4002
NAD 83 Zone Changes		
Nebraska	NE	2600

New England State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Connecticut	CT	0600
Maine East	ME_E	1801
Maine West	ME_W	1802
Massachusetts Mainlaine	MA_M	2001
Massachusetts Island	MA_I	2002
New Hampshire	NH	2800
New York East	NY_E	3101
New York Central	NY_C	3102
New York West	NY_W	3103
New York Long Island	NY_LI	3104
Rhode Island	RI	3800
Vermont	VT	4400

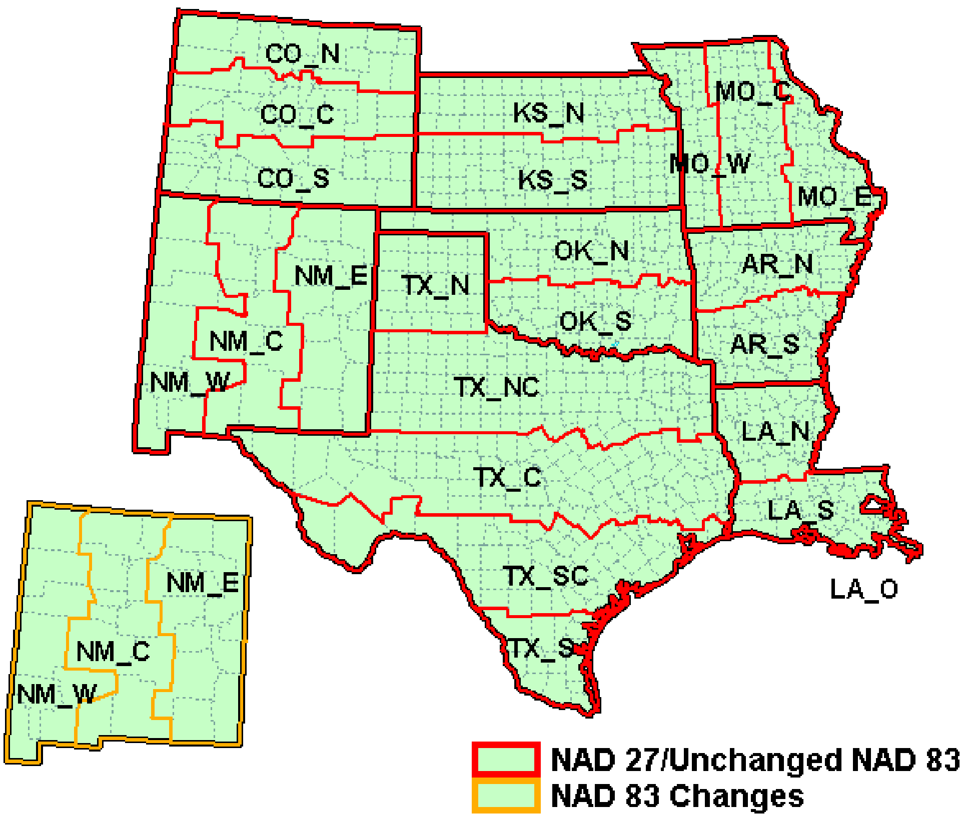
Northwest State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Idaho East	ID_E	1101
Idaho Central	ID_C	1102
Idaho West	ID_W	1103
Montana North	MT_N	2501
Montana Central	MT_C	2502
Montana South	MT_S	2503
Oregon North	OR_N	3601
Oregon South	OR_S	3602
Washington North	WA_N	4601
Washington South	WA_S	4602
Wyoming I	WY_E	4901
Wyoming II	WY_EC	4902
Wyoming III	WY_WC	4903
Wyoming IV	WY_W	4904
NAD 83 Zone Changes		

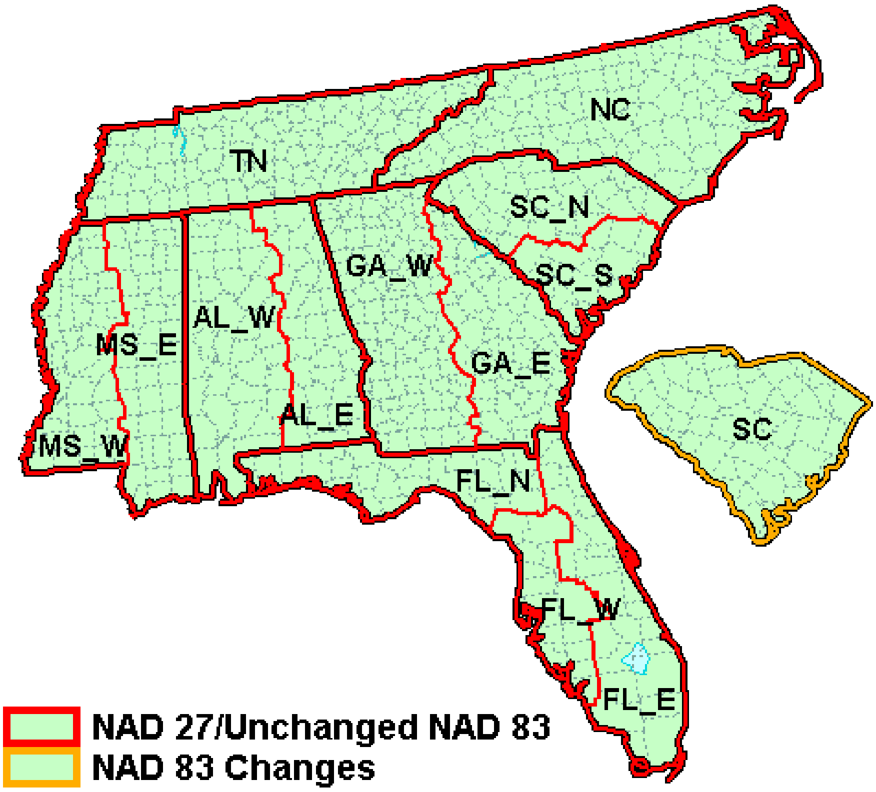
Montana	MT	2500
NAD 83 Name Changes		
Wyoming East	WY_E	4901
Wyoming East Central	WY_EC	4902
Wyoming West Central	WY_WC	4903
Wyoming West	WY_W	4904
NAD 83 Boundary Changes		
Washington North	WA_N	4601
Washington South	WA_S	4602

South Central State Plane



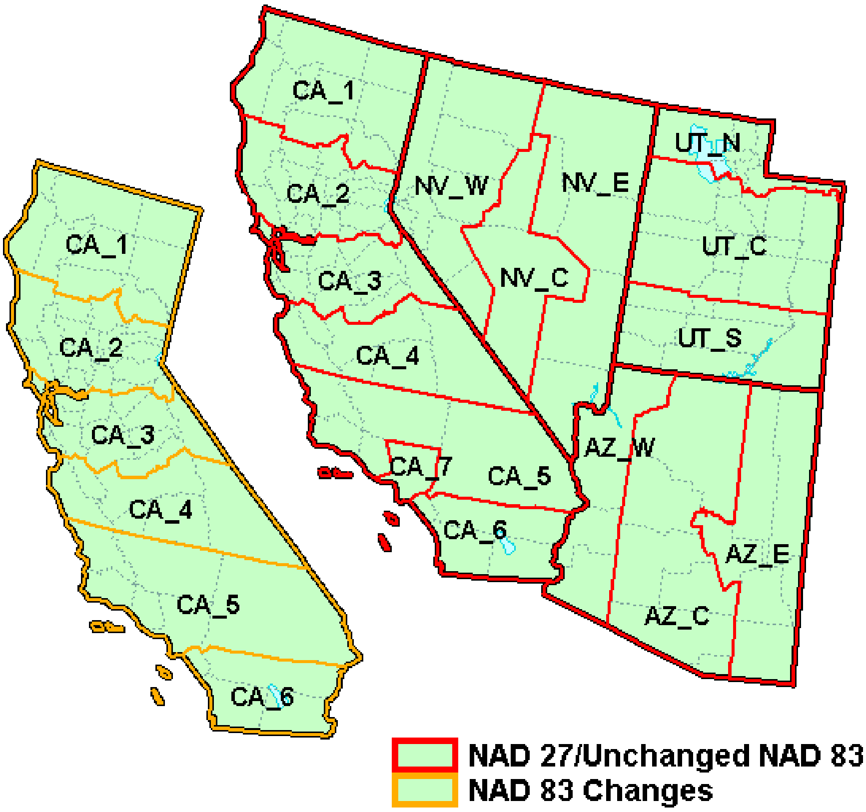
NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Arkansas North	AR_N	0301
Arkansas South	AR_S	0302
Colorado North	CO_N	0501
Colorado Central	CO_C	0502
Colorado South	CO_S	0503
Kansas North	KS_N	1501
Kansas South	KS_S	1502
Louisiana North	LA_N	1701
Louisiana South	LA_S	1702
Louisiana Offshore	LA_O	1703
Missouri East	MO_E	2401
Missouri Central	MO_C	2402
Missouri West	MO_W	2403
New Mexico East	NM_E	3001
New Mexico Central	NM_C	3002
New Mexico West	NM_W	3003
Oklahoma North	OK_N	3501
Oklahoma South	OK_S	3502
Texas North	TX_N	4201
Texas North Central	TX_NC	4202
Texas Central	TX_C	4203
Texas South Central	TX_SC	4204
Texas South	TX_S	4205
NAD 83 Boundary Changes		
New Mexico East	NM_E	3001
New Mexico Central	NM_C	3002
New Mexico West	NM_W	3003

South East State Plane



NAD 27 / 83		
Zone Name	**Map Code**	**Zone ID**
Alabama East	AL_E	0101
Alabama West	AL_W	0102
Florida East	FL_E	0901
Florida West	FL_W	0902
Florida North	FL_N	0903
Georgia East	GA_E	1001
Georgia West	GA_W	1002
Mississippi East	MS_E	2301
Mississippi West	MS_W	2302
North Carolina	NC	3200
South Carolina North	SC_N	3901
South Carolina South	SC_S	3902
Tennessee	TN	4100
NAD 83 Zone Changes		
South Carolina	SC	3900

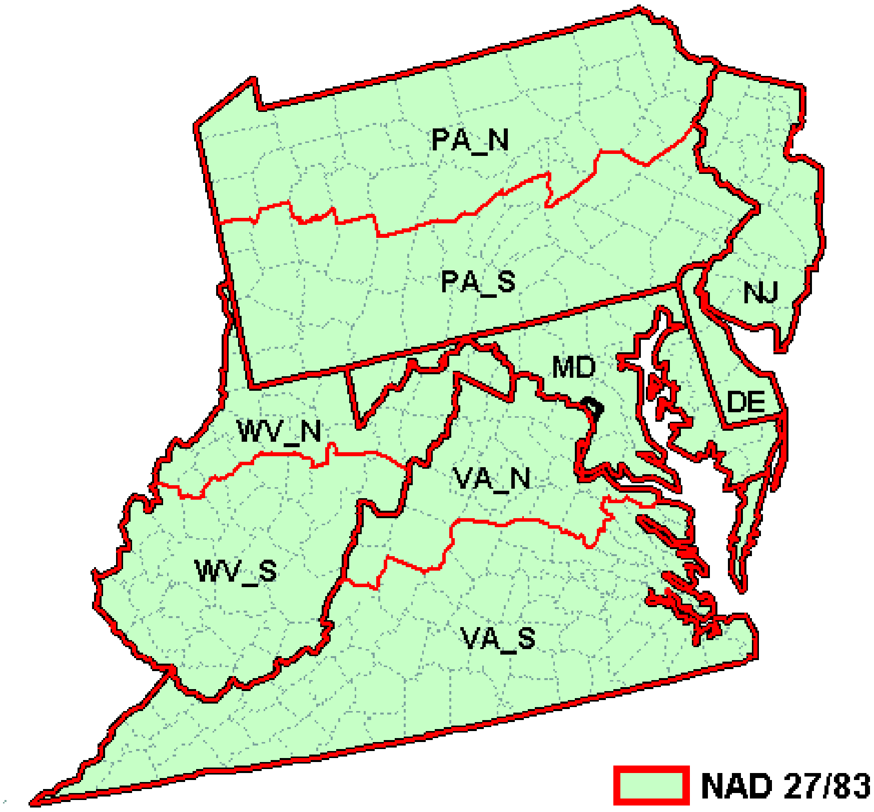
Southwest State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID

Arizona East	AZ_E	0201
Arizona Central	AZ_C	0202
Arizona West	AZ_W	0203
California 1	CA_1	0401
California 2	CA_2	0402
California 3	CA_3	0403
California 4	CA_4	0404
California 5	CA_5	0405
California 6	CA_6	0406
California 7	CA_7	0407
Nevada East	NV_E	2701
Nevada Central	NV_C	2702
Nevada West	NV_W	2703
Utah North	UT_N	4301
Utah Central	UT_C	4302
Utah South	UT_S	4303
NAD 83 Subtractions		
California 7	Removed	

Virginia Area State Plane



NAD 27 / 83		
Zone Name	Map Code	Zone ID
Delaware	DE	0700
District of Columbia / Maryland	MD	1900
New Jersey	NJ	2900
Pennsylvania North	PA_N	3701
Pennsylvania South	PA_S	3702
Virgina North	VA_N	4501
Virgina South	VA_S	4502
West Virginia North	WV_N	4701
West Virginia South	WV_S	4702

2.5. Datasets

Datasets

A dataset is a set of values associated with each node, cell, vertex, or scatter point in an object. A dataset can be steady state (one value per item, one time step) or transient (one value per item, multiple time steps). The values in the dataset can be scalar values or vector values. Certain types of objects in SMS have an associated list of scalar datasets and a list of vector datasets. Each of the following objects in SMS can have both scalar a vector datasets:

- Scattered Datasets
- 2D Meshes
- 2D Cartesian Grids
- Particle Sets
- 1D Grids

Datasets are used for both pre- and post-processing of models. For example, a scalar dataset associated with a 2D mesh can represent starting values of elevations or initial water surface elevations for a surface water modeling problem. Another dataset associated with the same mesh may represent computed water surface values. Datasets can be used to generate contours, vector plots, functional surfaces and animation sequences. The commands for manipulating datasets are located in the Current Model's Data menu.

Generating Datasets

Data sets can be generated in a variety of ways such as:

- Output from a surface water model (water level, velocity, concentration, transport, etc.)
- Tabular values in a text file entered by the user or exported from another application such as a GIS
- Created by interpolating from a scatter point set to a grid, or mesh
- Generated by performing mathematical operations on existing datasets with the Data Calculator

One advantage of the dataset approach for managing information is that it facilitates transfer of information between different models with differing resolution. This is accomplished through scatter sets and interpolation. Grids and meshes can be converted to a 2D scatter set. When an object is converted to a scatter set, all scalar datasets associated with the object are copied to the new scatter set. The datasets can then be transferred from the scatter set to other objects of any type using interpolation.

Right Click Menus

Datasets are displayed and managed in the Project Explorer. Right clicking on a data set invokes the right click menu which consists of a list of commands that can be performed on the dataset.

Dataset Information

The dataset info dialog allows the user to examine statistical properties of the dataset.

Active Dataset

Each module in SMS has a set of values designated as the "active dataset." The active data set is an important part of model visualization in SMS. Each time the display is refreshed, the contours and other display features are generated using the active dataset. Left-clicking on a solution or dataset in the Project Explorer makes that item "active". The icons used to identify the different datasets shown in the Project Explorer are as follows:

Dataset Type	Inactive Icon	Active Icon
Elevation		
Scalar		
Vector		

If the active dataset is transient then the time steps are displayed in the Time Step Window.

Solutions

Solutions are output from a numerical model that SMS supports. Solutions are shown in the Project Explorer as a folder. If a solution is transient then the time steps are displayed in the Time Step Window. The solution may contain text files such as the *.out and *.prt files produced by a model. These files can viewed by right clicking on the item and selecting **View File** from the pop up menu, or you can double click on the item.

Folder

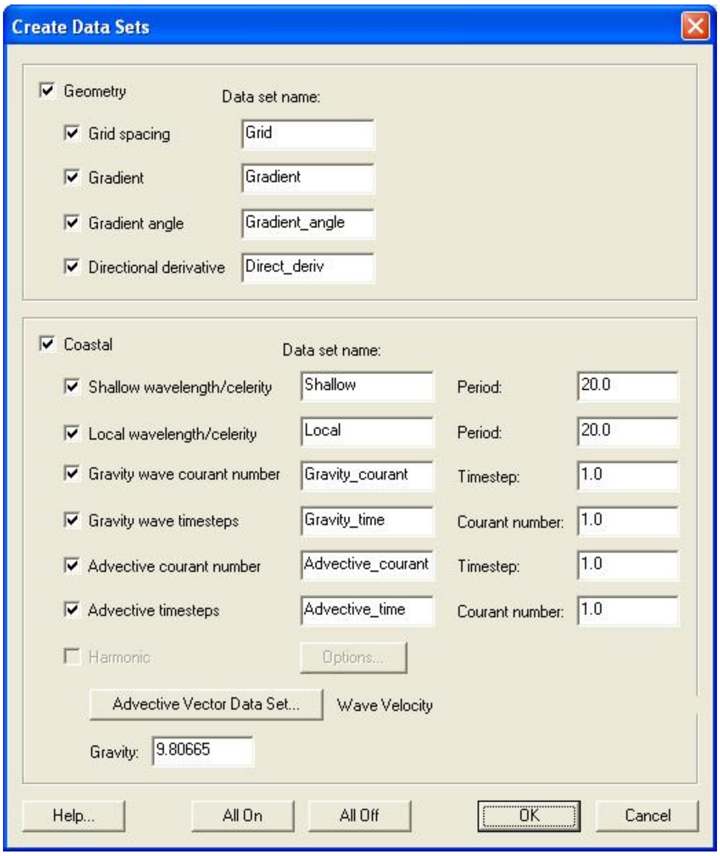
The datasets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the Project Explorer. Folders can be created by right clicking on the certain items in the Project Explorer and selecting **New Folder** in the menu. A dataset or folder can be deleted simply by selecting the folder and selecting the *Delete key* or by right-clicking on the item and selecting the **Delete** option in the corresponding pop-up menu.

Related Topics

- Layout of the Graphical Interface

Create Datasets

At a glance



Create Datasets

- Automatically generate commonly used datasets

The *Create Datasets* dialog is used to create functions for the entire mesh or active scatter set. The option is in *Data | Create Datasets* in both the Scatter and Mesh modules.

Each function that is toggled on will be created. All of the available functions can be turned on by pushing *All On*. All of the functions can be turned off by pushing *All Off*. The Gravity can be set and is used in several of the function calculations. The functions that can be created include:

- **Grid Spacing** – Creates a function that gives the average distance between a node and its neighbors.
- **Gradient** – Creates a function that gives the gradient at each node. The gradient is calculated at each point by averaging the normals of the faces connected to that point. For vertices in a TIN, this includes all the triangles connected to the vertex. For nodes in a mesh, it includes all the elements connected to the node. Currently no adjustment is made to account for the varied area of these faces. The gradient is defined as the run divided by the rise.
- **Gradient Angle** – Creates a function that gives the direction in degrees of the maximum gradient at each point. (See above for the method used to compute the gradient.)
- **Directional Derivative** – Creates a vector function that gives the gradient (run/rise) in the x and y directions. (See above for the method used to compute the gradient.)
- **Shallow Wavelength/Celerity** – Creates two functions that calculate the celerity and wavelength at each node in shallow water. The celerity is calculated as: Celerity = (Gravity * Nodal Elevation)^0.5. The Wavelength is: Wavelength = Period * Celerity.
- **Local Wavelength/Celerity** – Creates two functions that calculate the celerity and wavelength at each node for any depths.
- **Gravity Wave Courant Number** – Creates a function that gives the courant number for each node given the Time Step. The equation is: Courant Number = Time Step * (Gravity * Nodal Elevation)^0.5 / Nodal Spacing.
- **Gravity Wave Time Steps** – Creates a function that calculates the gravity wave time step given the Courant Number. The equation is the same as for the Gravity Wave Courant Number, solved for the Courant Number.

The final three functions can be created when the current numeric model is set to ADCIRC.

- **Advective Courant Number –** Creates a function that calculates the courant number given the Time Step and a velocity function. The velocity function can be selected by clicking on the Advective Functions&ldots; button. This brings up a *Select Dataset* dialog that lists the vector functions currently in memory. The courant cumber is calculated as: Courant Number = Nodal Velocity Magnitude * Time Step / Nodal Spacing. This option is disabled if no vector functions exist.
- **Advective Time Steps –** Creates a function that calculates the time step given the Courant Number and a velocity function. The velocity function can be selected as described above in the description of the Advective Courant Number. The equation is the same as for the Advective Courant Number, solved for the time step. This option is disabled if no vector functions exist.
- **Harmonic –** Creates a scalar harmonic function and/or a vector harmonic function. Pushing the Options button brings up the Harmonic Options dialog. The name of the function(s) to be created can be set in the Name fields. The frequencies to be used in creating the function can be chosen be double-clicking on a frequency name shown in the Scalar Frequencies window or be clicking on the name and by pushing Select. A frequency can be unselected by double-clicking on the name again or by selecting it and pushing Unselect. The time values that will be used in calculating the time steps can be set in the fields at the bottom of the dialog.

Related Topics

- Mesh Module Data Menu
- Scatter Module Data Menu
- Particle Module Data Menu
- Particle Module Create Datasets

Dataset Toolbox

The Dataset Toolbox contains numerous tools for working with datasets. Once the options for the current tool have been set and a name for the resulting dataset has been specified, selecting the Compute, Sample, etc. button will create the new dataset. The name of the new dataset will appear in the list of datasets.

The Dataset Toolbox tools are organized as follows:

Temporal

Sample Times

Create a new dataset from sampled times of an existing dataset. If "Interpolate times" is selected, linear interpolation will be used to determine the sampled times. If "Interpolate times" is not selected, the value from the nearest existing dataset time step will be used.

Derivatives

Create a new dataset of the change from one time step to the next, or the derivative from one time step to the next of an existing dataset. When computing a derivative, the time units must be specified. The the new dataset will output data in between the existing dataset time steps, resulting in one fewer time step than the original dataset.

Math

Compare

Compare two datasets by subtracting the "Alternate" dataset from the "Base" dataset. User specified NULL values are assigned if the base or alternate dataset is inactive.

Data Calculator

For more information, see Data Calculator.

Angle Convention

Create a new dataset with a different angle convention from a scalar dataset containing directions in a given angle convention. With datasets for CMS-WAVE and STWAVE cartesian grids, the angle can be converted to and from a shore normal convention.

Spatial

Geometry

- **Grid Spacing:** Creates a function that gives the average distance between a node and its neighbors.
- **Gradient:** Creates a function that gives the gradient at each node. The gradient is calculated as the run divided by the rise.
- **Gradient Angle:** Creates a function that gives the direction in degrees of the maximum gradient at each point.
- **Directional Derivative:** Creates a vector function that gives the gradient (run/rise) in the x and y directions.

Smoothing Datasets

For more information, see Smooth Dataset.

Conversion

Scalar to Vector

Converts two scalar datasets to a single vector dataset. The specified scalar datasets can be either magnitude and direction or x and y components.

Vector to Scalar

Converts a single vector dataset into two scalar datasets. The resulting scalar datasets can be either magnitude and direction or x and y components.

Coastal

Local wavelength/celerity

Creates two functions that calculate the celerity and wavelength at each node for any depths.

- **Celerity** = (Gravity * Nodal Elevation)^0.5.
- **Wavelength** = Period * Celerity.

Gravity Waves (Courant or Timesteps)

Creates a function that gives the courant number for each node given the Timestep, or the gravity wave timestep given the Courant Number.

- **Courant Number** = Timestep * (Gravity * Nodal Elevation)^0.5 / Nodal Spacing.
- **Timestep** = Courant Number * Nodal Spacing / (Gravity * Nodal Elevation)^0.5.

Advective (Courant or Timesteps)

Advective requires a vector function as input and is disabled if no vector functions exist. The courant option creates a function that calculates the courant number given the Time Step and a velocity function. The time step option creates a function that calculates the time step given the Courant Number and a velocity function.

- **Courant Number** = Nodal Velocity Magnitude * Timestep / Nodal Spacing.
- **Timestep** = Courant Number * Nodal Spacing / Nodal Velocity Magnitude.

Modification

Map Activity

This maps the activity array from one dataset to second dataset. This may be used to show only the values of interest on a particular dataset. This operation creates a new dataset.

Filter

This creates a new data set based on specified criteria. The following options are available for filtering:

- < (less than)
- <= (less than or equal to)
- > (greater than)
- >= (greater than or equal to)
- equal
- not equal
- null
- not null

If the value passes the specified filter, the following can be assigned:

- original (no change)
- specify (a user specified value)
- null (the data set null value)
- true (1.0)
- false (0.0)
- time - The first time the condition was met. Time can be specified in seconds, minutes, hours or days, and includes fractional values (such as 3.27 hours).

In addition, if the value passes none of the criteria, a default value can be assigned (see available options above).

The filtering is applied in the order specified. This means as soon as the new data set passes a test, it will not be filtered by subsequent tests.

Related Links

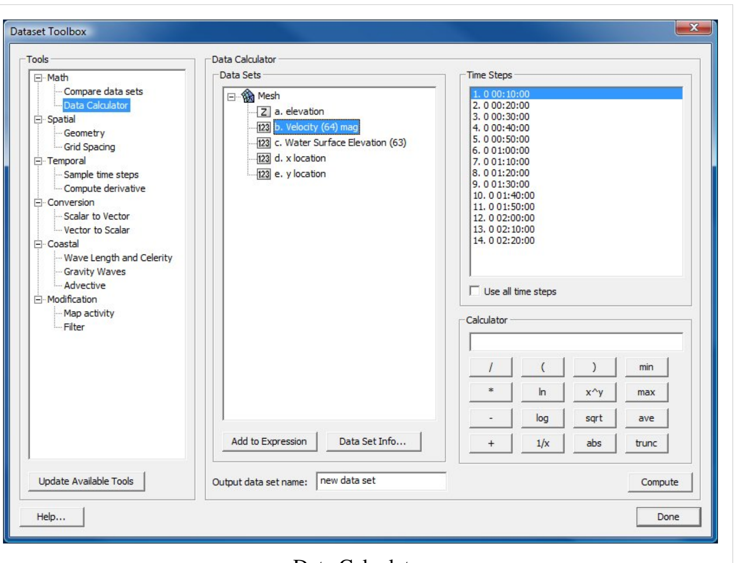
- Data Calculator
- Datasets

Data Calculator

At a glance

- Performs mathematical calculations on scalar datasets
- Calculations can include any number of scalar datasets and user supplied numbers
- Useful for computing derived values such as Froude numbers
- Useful for comparing scalar datasets

The Data Calculator can be used to perform mathematical operations with datasets to create new datasets. The Data Calculator is accessed by selecting the ***Data Calculator*** command from the *Data* menu. The components of the Data Calculator are as follows:



Data Calculator

Expression Field

The most important part of the Data Calculator is the Expression field. This is where the mathematical expression is entered. The expression should be formulated using the same rules that are used in formulating equations in a spreadsheet. Parentheses should be used to clearly indicate the preferred order of evaluation. There is no limit on the length of the expression. The operators in the expression should be limited to the operators shown in the middle of the Data Calculator. The operands in the expression should consist of user-defined constants (e.g., 3.14159), or datasets.

List of Datasets

All of the datasets associated with the active object (TIN, Grid, Mesh, or Scatter Point Set) are listed at the top of the Data Calculator. If a transient dataset is highlighted, the time steps are listed on the right side of the Data Calculator. When a dataset is used in an expression, the name of the dataset should NOT be used. Rather, the letter associated with the dataset should be used. For example, if a dataset is listed as "b. head1", the dataset is referenced in the expression simply as "b"

When a transient dataset is used in an expression, either a single time step or the entire sequence of time steps may be used. For example, the expression "abs(d:100)" creates a single (steady state) dataset representing the absolute value of the dataset at time = 100.0. However, the expression "abs(d:all)" creates a transient dataset representing the absolute value of each of the time steps in the original dataset.

Result Name

When an expression is evaluated, a new dataset is created and the name of the new dataset is designated in the Result field.

Operators

The allowable operators are listed in the middle of the dialog. Selecting one of the operator buttons adds the selected operator to the end of the expression. However, the operators can also be typed directly in the expression field. The function of each of the operators is as follows:

Operator	Function
" + "	Add
" - "	Subtract
" * "	Multiply
" / "	Divide
" ("	Left Parenthesis
") "	Right /Parenthesis
" log(x) "	The base 10 logarithm of a dataset
" ln(x) "	The natural logarithm of a dataset
" x^a "	(x) raised to the (a) power. (x) and (a) can be any mixture of constants and datasets
" abs(x) "	The absolute value of a dataset
" sqrt(x) "	The square root of a dataset
" ave(x,y) "	The average of two datasets
" min(x,y) "	The minimum of two datasets
" max(x,y) "	The maximum of two datasets
" trunc(x,a,b) "	Truncates a dataset (x) so that all values are >= a and <= b
" 1/(x) "	The inverse of (x) - Only available in SMS

Operating With Transient Datasets

Each argument in the operators listed in the table above may be:

- A steady state (1 time step) dataset
- A specified time step of a transient dataset (i.e., x:#). In this case the # represents the index of the time step as specified in the time step window.
- A transient time step (i.e., x:all). These operations are only valid if all arguments have matching time step values. In this case, the result will be a new transient dataset with identical time values as the arguments.

The data calculator supports an alternate format for computing attributes of a transient dataset. This alternate format applies to three of the operators. These operators compute a single time step (steady state) dataset representing the spatially varied attribute operating on all the time steps.

Operator	Function
" ave(x:all) "	The average at each location of all time steps in the dataset
" min(x:all) "	The minimum at each location of all time steps in the dataset
" max(x:all) "	The maximum at each location of all time steps in the dataset

Compute Button

Once an expression is formulated and a name for the resulting dataset has been specified, the expression can be evaluated by selecting the **Compute** button. At this point, the dataset is created and the name of the new dataset should appear in the list of datasets.

Related Links

- Datasets (GMS)
- Datasets (SMS)
- Datasets (WMS)
- Dataset Toolbox (SMS)

Size Datasets

A size dataset is used to determine how far apart to space the mesh nodes when:

- Generating a mesh using the Scalar Paving Density mesh types
- Relaxing a mesh

External Links

- Howlett, John David (2005). Size Function Based Mesh Relaxation. Thesis, Brigham Young University. [1]

Related Topics

- Smooth Dataset

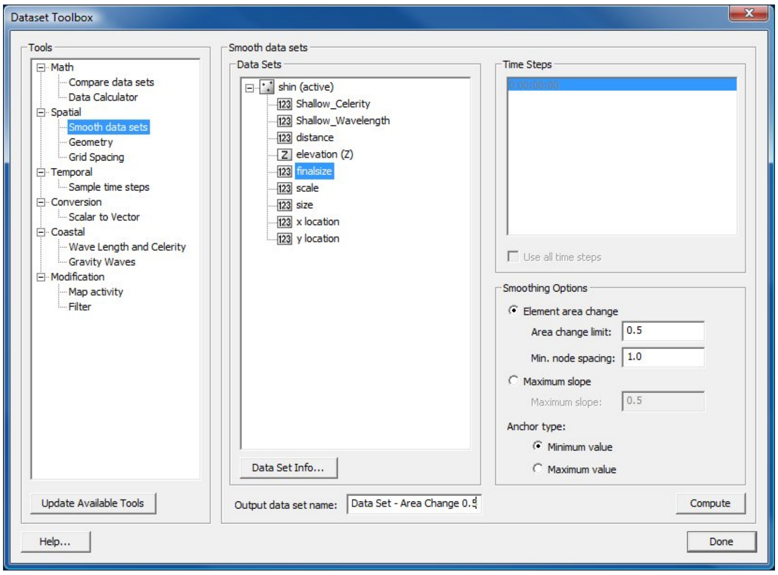
References

[1] http://contentdm.lib.byu.edu/utils/getfile/collection/ETD/id/287/filename/etd761.pdf

Smooth Datasets

The *Smooth Dataset* dialog is used to condition scattered data scalar values before those values are used in an interpolation process. This includes two general applications, smoothing a size dataset to prevent the dataset values from changing too quickly, and smoothing depth/elevation values to prevent extreme slopes.

The *Smooth Dataset* dialog is accessed via the Dataset Toolbox by selecting *Data* | *Dataset Toolbox* in the Scatter module.



Smooth Datasets Dialog

Smoothing Size Datasets

One measure of mesh quality is element area change. If the dataset values change too quickly in a size dataset, the element area change of adjacent elements may be too great, resulting in poor mesh quality.

Smoothing options

- **Element area change limit** – The selected dataset values will be modified to honor the specified element area change limit. This value defines the maximum ratio between adjacent points based on the distance between points.
  - **Minimum value anchor type** – Dataset values are decreased. Results in a more refined (more nodes/elements) mesh when used as a size dataset.
  - **Maximum value anchor type** – Dataset values are increased. Results in a less refined (fewer nodes/elements) mesh when used as a size dataset.
- **Minimum node spacing** – The minimum value allowed in the smoothed dataset.

Tips

- After smoothing a size dataset, use the data calculator to subtract the smoothed sized dataset from the original dataset and create a "change" dataset. Contour the "change" dataset to easily determine what and where changes were made by the smoothing algorithm.

Smoothing Elevation/Depth Datasets

This option allows the user to specify a maximum slope. The process creates a new data set which honors the maximum specified slope.

- Minimum value anchor type – The smoothing operation anchors the minimum data set value (such as the lowest elevation or smallest depth) and adjusts the adjacent values to ensure the slope is less than or equal to the specified slope.
- Maximum value anchor type – The smoothing operation anchors the maximum data set value (such as the highest elevation or largest depth) and adjusts the adjacent values to ensure the slope is less than or equal to the specified

slope.

The operation includes all scatter points if none are selected. Alternatively, the user may select a group of scatter points to be operated on. Points not selected will not have their scalar value modified. This means the only way to modify a point is if it has an adjacent point that is also selected and the slope between these two selected vertices is steeper than the maximum specified slope.

Related Topics

- Scatter Data Menu
- Size Dataset
- Relax Elements
- Dataset Toolbox

Metadata

Metadata, or data about data, can be crucial in the modeling process. In many situations, metadata is saved in a separate file (called a metadata file). A common problem is that metadata and the data it describes are often separated. Metadata is of little value without the data files it relates too. At the same time, metadata makes the data more usable and therefore, more valuable.

In the SMS, metadata can be cataloged inside the project file. Metadata can be associated with the project as a whole, a single geometric object such as a survey or finite element domain, or individual components down to the dataset level.

Project/Geometric Object Metadata

The project metadata can be accessed through the *Edit|Metadata...* command. Object metadata can be accessed by right clicking on the object and choosing the *Metadata...* command. Either method invokes the *Metadata* dialog which includes edit fields for the following metadata:

- Title of the project or object
- Abstract – a brief description
- Purpose – this can change over time, but if it is recorded with the project when generated, it provides a valuable backdrop when applying the project for other purposes at later times.
- Creation Date – This is filled in automatically to the current date when the project is created. It can be edited if an existing project is simply being organized.
- General Topics – This section includes a list of topics and description or blurb for each topic. As the project develops or is modified, notes of these developments, consisting of dates, individuals involved, purposes, etc, can be annotated to the project.
- Profile – this button invokes the "Profile Information dialog" which documents who created this project and provides information about this individual. The profile can be set up and associated with an installation of SMS and then this information is automatically added to all projects created with this installation. In order to associate a profile as the default, simply click the "Use Current for Default" button in the Profile Information dialog.
- Spatial – this button invokes a dialog displaying the projection used by the project and the spatial limits of the project. This information is automatically filled in.
- Source – this button is only available for objects (not the project as a whole). It brings up a the Source Metadata dialog. Whenever SMS generates a new object, such as a mesh from a conceptual module, or reads in a new object from a file, the source is recorded. In the former case, the coverage and scatter set used will be recorded. In the latter, the filename. This dialog allows the modeler to record additional notes about this object.

Dataset Metadata

Dataset metadata is accessed by right clicking on the dataset. This invokes the *Dataset Metadata* dialog which includes a text string describing the dataset.

Dataset Info

A dataset consists of a set of values. It is often useful to be able to get general information about the dataset. SMS displays this information in the *Dataset Info* dialog which is invoked by right clicking on the dataset and selectiong the *Info...* command.

The dialog displays the following information:

- Name – display the name of the dataset
- Number of time steps – displays the number of discrete times represented by the dataset. It will be 1 for a steady state dataset.
- Beginning time – this shows the lowest time for any discrete time represented by the dataset. For steady state datasets, this will be 0.0. It will be displayed in the information dialog according to the current time settings.
- Ending time – this shows the largest time for any discrete time represented by the dataset. For steady state datasets, this will be 0.0. It will be displayed in the information dialog according to the current time settings.
- All time steps statistics – this displays the minimum, maximum and range of values for all values in the dataset across all times represented by the dataset.
- Current time step statistics – this displays the minimum, maximum, range, mean and standard deviation of values for all values in the current or actively selected time.
- Reference Time – If a dataset is referenced to a universal date/time, it often will be stored relative to a "zero time". SMS allows the user to specify a "zero time" for a project that will be applied to all datasets that do not have their own reference. This field displays the "zero time" for this dataset if it exists.

Related Topics

- Datasets

2.6. Display Options

Display Options

Display Options in SMS refers to the control of what entities are displayed, and how (color and style) they are displayed. Each entity in each module has its own display options. The display options for the active module are shown when the Display Options dialog opens.

The Display Options dialog is opened by any of the following methods:

- Right clicking on the Module type folder in the Project Explorer and selecting the Display Options command from the right click menu
- Using the Display | Display Options menu command
- Clicking the  display options Macro
- Using the keyboard shortcut *CTRL+D*

Display Option Pages

SMS supports a display option page for general display options and for each type of data (i.e. 2D mesh, scatter sets, map data, ...) managed in a simulation. The *display option* dialog includes the option to set all options on all pages if desired. The toggle at the lower left of the dialog hides the option pages for data types that are not included in the current project. This toggle is selected by default and reduces the amount of information a user sees. The following display option pages exist:

- 1D Grid: This page controls the display options related to the coastal morphology model GenCade.
- 2D Mesh: This page controls the display options related to unstructured 2D meshes and the models (such as ADH and the generic model interface) that use them.
- Cartesian Grid: This page controls the options related to Cartesian grids and the models (such as TUFLOW, CMS-Flow and BOUSS2D) that use them.
- Curvilinear Grid: This page controls the options related to the curvilinear or boundary fitted grids and the models (such as LTFATE) that use them.
- General: This page controls the general display options.
- GIS: This page controls the display options related to GIS data using the internal GIS support in SMS. If ArcGIS is enabled inside of SMS, the ArcObjects display options control the display of these entities.
- Map: This page controls the display options related to the map module (coverages) in SMS.
- Mesh: This page controls the display options related to the general unstructured mesh objects.
- Particle: This page controls the display options related to the PTM Lagrangian particle tracker model.
- Raster: This page controls the display options related to raster (or DEM) type objects.
- Rivhyd: This page controls the display options related to one dimensional river models.
- Scatter: This page controls the display options related to scattered datasets (also referred to as Triangulated Irregular Networks or TINs).

Tabs

For each page, additional tabs may appear to allow the user to specify the display settings for vectors or contours in that type of data. The General Display Options also includes separate tabs for Lighting Options and viewing control.

Functional Surfaces

For some types of data, it could be useful to display a surface of the datasets associated with the geometric data. For example, a computed water surface can be displayed as a surface to intuitively illustrate how a flooding scenario looks in three dimensions. Other datasets, which don't have such a direct connection to elevation can also be viewed as a functional surface to give insight to the situation. The options to set the functional surface for a specific geometric data type will appear in the page for that data type.

Entity Display Options

For each entity type, the dialog includes a toggle, and where appropriate, a button. If the toggle is selected, SMS will display the entity type. The buttons are of various types, as described below:

Points

If the display of an entity is focused around a single location, like a mesh node, the button displays a circle, drawn in the color that will be used to display that entity. To change the circle size, or color, the user can click on the button, and the point attributes dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button. It is recommended that if there are many of points, and their position can be inferred from other displayed entities (such as the position of mesh nodes by the edges of the elements) that the symbols be turned off to increase efficiency.

Symbols

If the display of an entity is focused around a single location, like a scatter point, the button displays a symbol, drawn in the color that will be used to display that entity. To change the symbol, symbol size, or color, the user can click on the button, and the Symbol Attributes dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button. It is recommended that if there are many of these entities, and their position can be inferred from other displayed entities (such as the position of scatter points by the edges of the scatter triangles) that the symbols be turned off to increase efficiency.

Line

If the entity to be displayed encloses a region, such as a triangle or element, the button displays a sample line drawn in the color and width that will be used to display the line around the edge of that entity. To change the color and/or width, the user can click on the button, and the Line Attributes dialog will appear. This allows the user to select a line style (dashed or solid), a width (in pixels) and a color. The color can also be changed by clicking on the combo box arrow next to the button.

Font

If the entity to be displayed is a text string, such as the node id, the button displays a sample string ("AaBb") drawn in the color and font that will be used to display the string. To change the font, the user can click on the button, and the Font dialog will appear. The color can be changed by clicking on the combo box arrow next to the button.

Color

If the entity has only a color associated with it, the button displays a square, drawn in the color that will be used to display that entity. To change the color, the user can click on the button, and the Color dialog will appear. The color can also be changed by clicking on the combo box arrow next to the button.

Options

Other specific display attributes can be accessed through the *Options* buttons. The options available will vary based on the current model and includes attributes such as boundary conditions assigned to nodes or nodestrings.

3D Visualization

Post-Processing tools inside SMS help to visualize the model solution created by running the simulation through the solver. Post-processing tools available on solution data vary based on the module but include:

- 2D Plots
- Animations
- Contours
- Calibration Targets
- Vector Visualization

Related Topics

- General Interface Features
- Color Ramps
- Contour Labels
- Display Options

Color Options

The *Color Options* dialog lets the user determine how the contours and vectors will be colored. The *Color Options* dialog is opened from the Data menu, or from the **Color Ramp...** button on the Contours and Vectors tabs of the Display Options dialog.

The default color method is the *Solid* color option. This method uses a single color for all contours. As an alternative, the user can define a ramp of colors. These colors are distributed across the range of contour values in a continuous fashion, giving each contour its own color. If specific contour values are specified, the user controls whether the colors are distributed by index or by value in the upper right portion of the dialog. The following types of color ramps are supported by SMS:

- Solid color – A single color is used for all values.
- Intensity ramp – The color ramp is defined as a continuous variation of the intensity of the default solid color. This is the same color used for the Solid color option.
- Hue ramp – The ramp is a continuous variation of hues using the hue-saturation-value color model.
- User Defined Palettes – User defined color ramp.

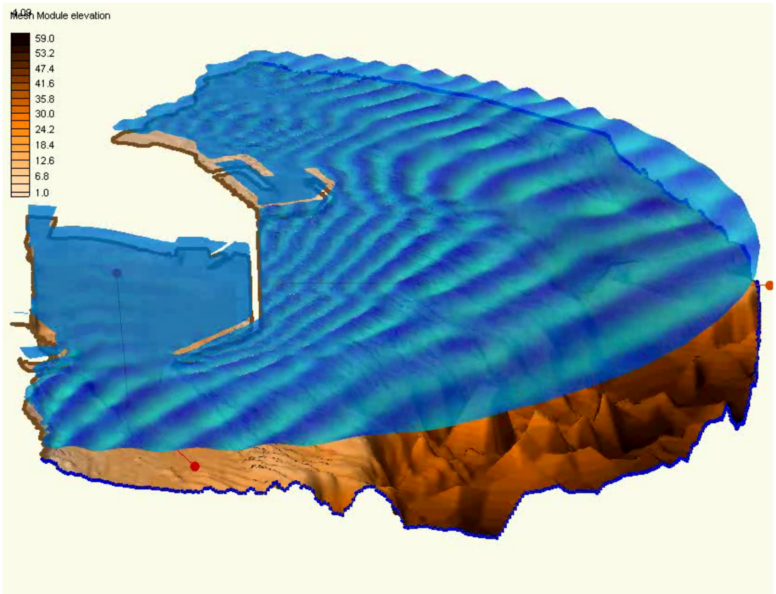
If using an intensity ramp or hue ramp, the ramp can be edited to include only a portion of the entire ramp, or converted to a User Defined Palettes for further editing. The user modifies the portion of the ramp to be used by setting the minimum and maximum values for hue or intensity with the scroll bars in the Color Options dialog. These controls specify where the minimum value will be mapped into the ramp and where the maximum value will be mapped. The reverse button changes the direction of the color gradation in the color ramp.

Related Topics

- Contour Options
- Contour Labels
- Display Options
- Vector Visualization
- User Defined Palettes
- New Palette Dialog

Functional Surfaces

At a glance



Wave functional surface

- Surface with elevation based upon scalar dataset values
- Very useful for wave models and models with large change in water surface elevation
- Elevations can be exaggerated to better visualize dataset variations
- Surfaces can have a solid color or use color filled contours
- Transparency can be used to allow see through surfaces

A functional surface is exactly that. It is a surface representing one of the functional datasets associated with a mesh, grid or TIN. The most intuitive example of a functional surface is the display of the water surface over a model's bathymetry. In this case, the surface represents an actual physical surface, but the functional surface could just as easily represent the velocity magnitude, or concentration, or any other scalar quantity.

To create/display functional surfaces, the user must enable them in the display options of the appropriate module, and specify their attributes which include:

- **Dataset** – the user selects which dataset is to be used to form the functional surface.
- **Z Offset** – SMS displays functional surfaces at a simulated z-value. This may be the actual surface value (such as is the case with water surfaces elevations), but more often the value will not have a physical meaning, and may intersect the bathymetry or not even be in the same area. For this reason, SMS offers options for placing the functional surface at its real values, relative to the bathymetry, or at a user specified offset.
- **Z Magnification** – Functional data may not vary significantly when compared to the horizontal extents of the model. For this reason, the interface allows magnification (scaling) of the functional surface. By default, the surface is scaled based on the global z-magnification specified in the general display options. This may be overridden.
- **Display Attributes** – the user controls the color of the functional surface. It may be a constant color or colored based on the contour colors specified. The colors may be associated with the value of the functional surface, or another dataset. The surface may also be partially transparent.

Related Topics

- Cartesian Grid Display Options
- Mesh Display Options

Lighting Options

This dialog allows the user to control the shading of faces in the SMS display. By default, all objects are displayed in the color specified by their attributes. However, objects such as elements, cells and triangles which cover an area, can be more intuitively understood if they are shaded as a three dimensional entity. The shading options includes two toggles, one slide bar and a light position window.

The lighting options are accessed by clicking on the Lighting Options item or tab in the *Display Options* dialog. The default options vary between applications, and the options may be changed, saved, and restored within the project.

Toggles

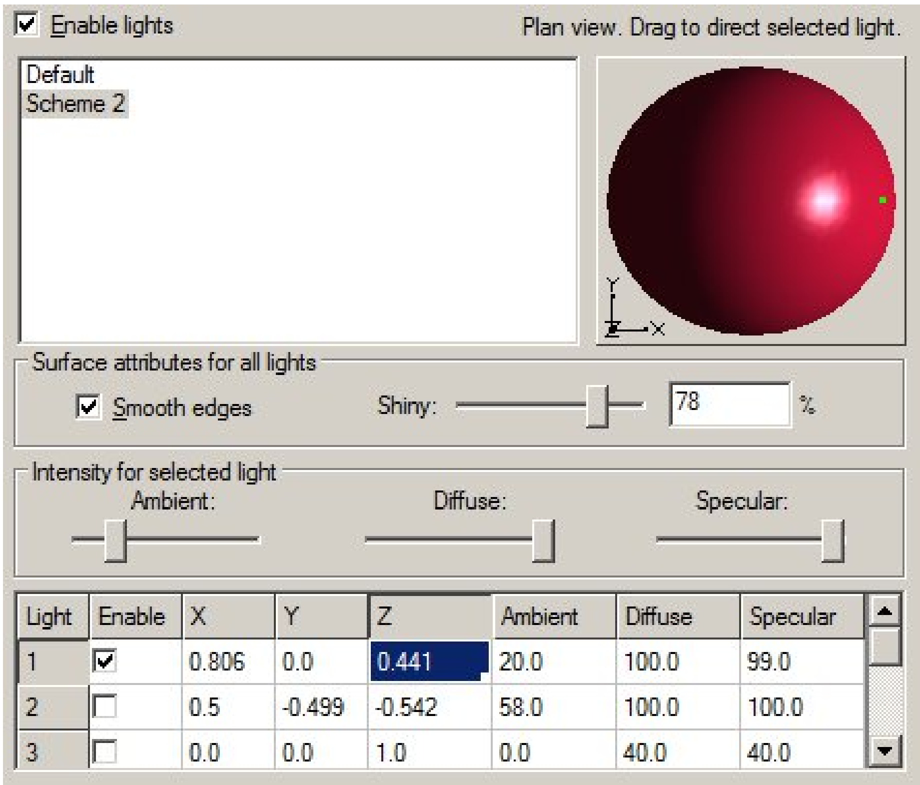
The first toggle allows the user to turn on the use of a light source. When this toggle is selected, the second toggle becomes available. The second toggle tells SMS to smooth corners between adjacent faces. This allows the faceted surface to appear as a smooth surface.

Slider

The slide bar allows the user to specify the amount of ambient light. Ambient light is the minimum intensity (brightness) to be displayed. A recommended value is between 0.2 and 0.4.

Light Position

The right side of the dialog allows the user to set the light direction and gives a preview of that direction displayed on a sphere.



The following table describes the lighting display options.

Display Option	Description
Enable lights	This check box controls whether light sources are used in the lighting process for generating lighted images. These light sources control the intensity of the colors on the lighted image and highlight the relief or geometrical variation in the surface of the objects being lighted.
Lighting list box	This list contains preset lighting schemes and highlights the scheme currently displayed.
Renaming a scheme	Double click on a scheme to begin editing its name.
Deleting a scheme	Right click on a scheme and select Delete. The final scheme cannot be deleted.
Creating a scheme	Right click on a scheme and select duplicate.
Plan view preview	This preview shows the current light scheme on a sphere in plan view, i.e., looking along the z-axis. Click or drag within the preview to direct both the diffuse and the specular components of the light currently selected in the light table. The selected light direction is shown by a dot on the sphere. A direction from in front of the sphere is shown by a green dot, and from in back by a red dot.
Smooth edges	Check this box to smooth all diffuse and specular lights of this scheme so that the surface does not appear faceted.
Shiny	Increase this value to sharpen all specular highlights of this scheme. At 100% this value turns off the specular highlight since it assumes that all specular lights are points whose reflection shrinks to a imperceptible point at maximum shinyness. At 0% this value assumes that the full intensity of the light is reflected in all directions (decrease the specular values proportionally to get a realistic effect of less and less light reflecting to the eye from each surface).
Ambient slider	Shows the Ambient value of the light currently selected in the table, and can change the value. The ambient value is light from all directions which lights each and all surfaces uniformly leaving no surface unlighted. It is most useful on surfaces facing away from directional light such as diffuse and specular light.
Diffuse slider	Shows the Diffuse value of the light currently selected in the table, and can change the value. The diffuse value is for a point light which brightens surfaces in all directions the more they face the that light, and which leaves surfaces in darkness that face away from the light.
Specular slider	Shows the Specular value of the light currently selected in the table, and can change the value. The specular value is a point light which brightens surfaces if they reflect like a mirror from the direction of the light to the direction of the viewer, and which leaves surfaces in darkness that do not have this angle of reflection.
Light table	Displays the enable, xyz position, Ambient, Diffuse, and Specular values for each of 8 lights in the current scheme, and highlights the currently selected light. Any of these values may be modified by clicking them and editing their value.
Enable column	Check these boxes to turn on each light.
X, Y, and Z columns	Edit these values or click/drag in the plan view preview sphere to change the direction of the light. These values are will be normalized to a unit direction vector.
Ambient, Diffuse, and Specular columns	Edit these values or drag their corresponding slider.

Raster Options

Importing Rasters

Import a raster file by selecting Open in the File menu. Select the proper raster file as shown in the following table. Select Open. At the popup "Load it as...", select DEM.

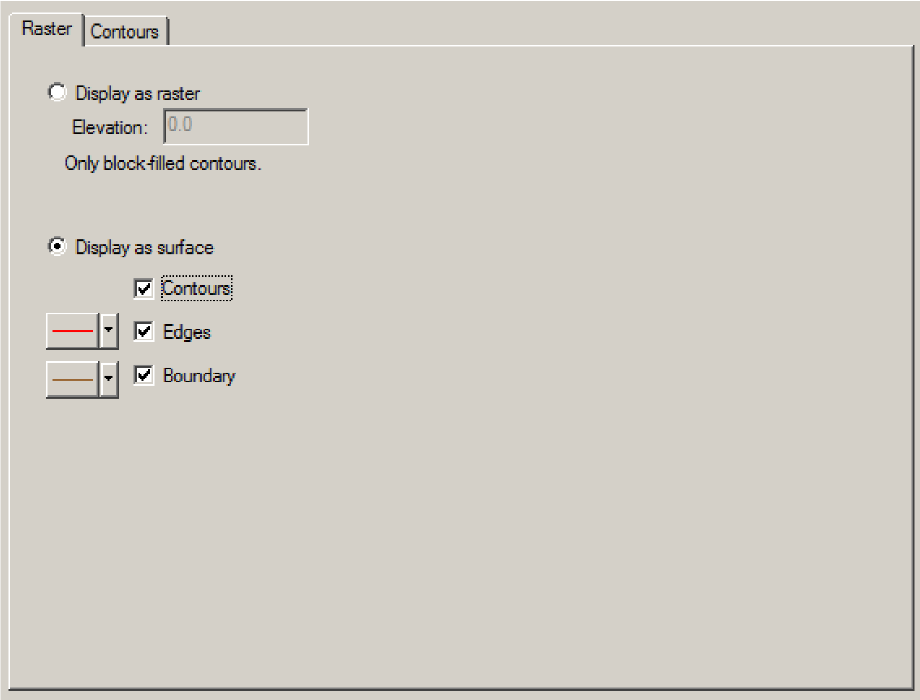
Format	File to Open	Source	Importance	Level of Support
ArcInfo Binary Grid	w001001.adf	ESRI	1	Supported in gms and in sms
ArcInfo Ascii Grid	asc	ESRI	2	Supported in gms and in sms
USGS DEM Grid Float	flt	[1] in gms and in sms		
USGS NED Grid Float	*.flt	[1] in gms and in sms		
Canadian DEM	*.dem	[2] in gms and supported in sms		
DTED	*.dt0	ERDC	3	Supported in gms and supported in sms
Aster DEM	*.tif	[3] in gms and supported in sms as images		
SDTS	*.ddf	[4] in gms and supported in sms		

How to export TINs in adf format from ArcGIS in a format that XMS will read

1. Load the TIN into ArcMap
2. Expand the “3D Analyst Tools | Conversion Tools | From TIN” toolset in ArcToolbox
3. Double click the “TIN to Raster” tool in ArcToolbox (specify your current TIN file in the "Input TIN” field). Make a note of the path in the "Output Raster" field.
4. Expand the “Conversion Tools | From Raster” toolset in ArcToolbox
5. Double click the “Raster to ASCII” tool in ArcToolbox (specify the raster file that you created in the previous step as the input raster, and make a note of the path for the output file)
6. Open Windows Explorer (My Computer) and browse to the location of the ASCII .txt file output in step 5
7. Make a copy of the *.txt file created in step 5
8. Change the extension of the *.txt file to *.dem
9. Open the “.dem” file in WMS

Displaying Rasters

The raster display options are accessed by clicking on the Raster Options item or tab in the Display Options dialog. The default options vary between applications, and the options may be changed, saved, and restored within the project.



The following table describes the raster display options.

Display Option	Description
Image Display	Select the "Display as raster" radio button to display the raster as a flat image rather than as a surface with elevation changes. Contour options are applied to form the image with block color fill.
Image Elevation	The raster image is drawn at an elevation of 0.0 by default. Change the "Elevation:" value to draw it at a different elevation.
Surface Display	Select the "Display as surface" radio button to display the raster as a height varying surface rather than as a flat image. Enable either Contours, Edges, or Boundary to see that type of surface or nothing will be shown.
Surface Contour	Select the "Contours" check box to apply contour options to the surface with contour lines and/or smooth color fill.
Surface Edges	Select the "Edges" check box to display the polygonal edges between height samples in the surface. The control to the left sets line color and either enables line dashes or species line width for the edges. This is typically the slowest surface to render.
Surface Boundary	Select the "Boundary" check box to display only those polygonal edges between height samples on the perimeter of the surface. The control to the left sets line color and either enables line dashes or species line width for the boundary. This is typically the fastest surface to render.

Back to XMS

References

[1] http://seamless.usgs.gov||2||Supported
[2] http://www.geobase.ca||3||Supported
[3] http://asterweb.jpl.nasa.gov/gdem-wist.asp||4||Supported
[4] http://data.geocomm.com/dem/demdownload.html||5||Supported

General Display Options

The general display options control display of general graphical control. It includes three tabs including:

General Tab

- **Drawing Options:**
  - Z magnification
  - background color
  - erase behind labels
- **Triad:**
  - SMS can display a coordinate triad at the lower left of the screen to display the orientation of the data in the display window. The user can control the size and color of this triad.
- **Texture mapping:** Currently SMS supports images displayed in the background and texture maps draped over TINS, grids, and meshes.
- **Drawing Grid:** SMS can display a grid (in plan view) behind all data on the graphics window.
  - Grid Spacing: The user can enter the increment between grid points. Remember that the grid can be used for both snapping and display, and not all grid lines need to be displayed.
  - Snap to Grid: If this toggle is on, newly created points, nodes and vertices are moved to the nearest point on the grid.
  - Display grid lines every n spaces: This option allows the user to specify how many grid lines to between displayed grid lines. The line style is also selected.
  - Display grid points every m spaces: This option allows the user to enable the display of a point at selected intervals along with the symbol attributes for the points.

Shading Tab

The shading tab allows the user to access to the Lighting Options in SMS.

View Tab

The view tab in the general display options allows the user to see/edit the current view parameters. This includes the specification of the type of view (plan or 3D) and the range of the data that is displayed on the screen.

Z Magnification

Justification for Z magnification

In most situations simulated in SMS, the data range in the horizontal direction is not similar to the data range in the vertical direction. For example, when simulating a river reach, the river may cover miles (or kilometers) along the length of the river, but in the z-direction, the change in elevation will only be in the tens to hundreds of feet (meters). In an opposite situation, when working with a coastal circulation model in geographic coordinates, the horizontal variation of the data may only be a few degrees, while the vertical change in depth can be thousands of meters. When displaying data in plan view, this inconsistency of data ranges does not cause a problem. However, when attempting to view data in an oblique view (from an angle in three dimensions), the first case of a long river ends up looking like a flat plane while the second case is just a mass of vertical bumps. To allow for intuitive display of the data in three dimensions SMS allows the specification of a Z magnification term. This scale factor exaggerates or reduces the relief of the data in the simulation.

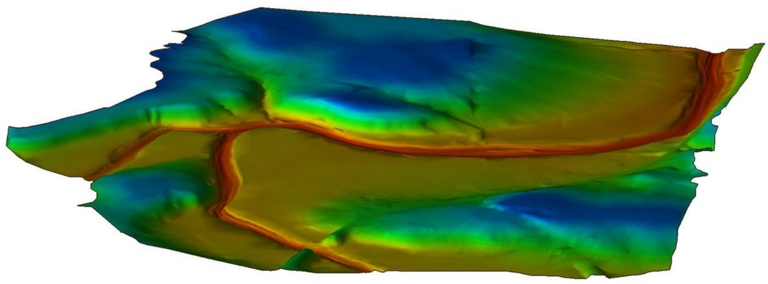
Auto Z magnification

SMS also includes the option to compute a Z magnification term automatically. This option is turned on by default. This means that every time SMS frames the data in a display, the Z magnification term is computed to ensure that the scaled span of the vertical data is just under 10% of the horizontal data. This prevents the data from becoming too flat (unless it is totally flat) and prevents the relief from becoming to drastic or dramatic. Since the Z magnification value is computed when SMS frames the data, modifications to the data that change the Z range are not incorporated into the magnification value until a frame command is encountered. This may result in difficulties rotating a scene in three dimensions. Operations like generating new elevation data for a mesh by interpolation from a raster or scatter set may cause this to occur. The user can disable the auto Z magnification feature by unchecking the toggle next to the "Auto z-mag". When the toggle is unchecked, an edit field appears allowing the user to specify a Z magnification value. The value is set to the previously computed auto Z magnification.

Contour Options

At a glance

- Visualize scalar datasets
- Linear, color filled or both at the same time
- Variable level of transparency
- Full control of ranges and colors
- Precision control for labels and legends



3D view of color filled contours

SMS can generate contours from a scalar dataset. Contour display is enabled using in the Display Options dialog. Unique contour display options can be set for each module that uses scalar datasets. Contour options can also be set for an individual dataset.

The module contour options can be edited by:

- The menu command *Data* | *Contour Options*. The data menu is available in most modules. See Module Specific Menus for more information.
- Opening the Display Options dialog
- The *Contour Options Macro*

The dataset specific contour options for an individual dataset can be edited by:

- Right click on the dataset in the Project Explorer and choose "Dataset Contour Options". This will open a dialog where you can define dataset specific contour options. If you want to go back to using the module contour options, right click on the dataset in the Project Explorer and choose *"Clear Dataset Contour Options."*

Data Range

The name, minimum value, and maximum value for the active time step of the dataset are shown. These values are sometimes useful when choosing an appropriate contour interval.

A minimum and maximum contour value can be specified, restricting the contours interval which will be shown. If the range is not specified, SMS will automatically choose a range based on the minimum and maximum value for the active time step.

Contour Interval

The contour interval is user controlled. Options include:

- *Number of Contours* – Specify the number of contours to use. SMS will automatically determine the value for each interval based on the specified range or dataset range.
- *Specified Interval* – Specify the interval to use for contours. SMS will automatically determine the number of contours needed based on the specified range or dataset range.
- *Specified Values* – Specify the number of contours to use and interval.

The items in the upper right section of the *Contour Options* dialog control the display of a contour legend and the option to accentuate some of the contours. If the *Show Color Legend* option is selected, and the contours are not being displayed as a single color, a legend of colors and corresponding data set values is displayed in a corner of *Graphics Window*. For color filled contours, this legend is a vertical strip of colors with text labels for the contour levels. If the contours are being displayed as linear segments or cubic splines, the legend is displayed as a series of contour level values and a line drawn in the color corresponding to that level. The size, location, label and font for the legend are set using the *Legend Options* dialog. If the user enters the title "DS" for the legend title, the name of the current dataset is used. If the user enters "DS:TS" the current dataset and time step are used as the title.

The options in the middle of the right side of the dialog control how the contours are computed and displayed. Three contouring methods are available:

- The default method is *Normal Linear Contours* and causes the contours to be displayed as piece-wise linear strings.
- If using the *Color* fill between contours method, the same linear contour strings are computed, but the regions between adjacent contour lines is filled with a solid color.
- If using the Cubic Spline Contours method, the contours are computed in strings and drawn as cubic splines. Drawing the contours as splines can cause the contours to appear smoother. Occasionally, loops appear in the splines or the splines cross neighboring contour splines. These problems can sometimes be fixed by adding tension to the splines. A tension factor greater than zero causes the cubic spline to be blended with or converge to a linear spline based on the same set of points. A tension factor of unity causes the cubic spline to coincide with the linear spline.

In the lower right corner of the *Contour Options* dialog, two buttons allow the user to specify the contour colors and the contour labeling options.

Contour Labels

The Contour Label Opts command in the **Data** menu is used to access the *Contour Labels Options* dialog which can be used to set the label color, font, spacing, size, etc. The dialog may also be invoked through the *Contour Options* dialog.

Labels can be added to contours one of two ways:

1. The upper left portion of the *Contour Label Options* dialog controls the generation of automatically spaced contour labels. The user can toggle the generation of automatic contour labels on or off. If the toggle is on, the user also specifies which contours should be labeled and the distance along the contour between labels.
2. In some modules, contour or function labels can be added manually to an image by selecting the Contour Labels tool in the Tool Palette and clicking on the mesh or grid where a label is desired. If the Place on contours option in the upper right portion of the *Contour Label Options* dialog is selected, the label is moved to the closest contour and the contour is labeled there. If the Place under cursor option is selected, the label shows the value of the point at the click location and is placed there. This option is useful to post data set value labels in regions where there are no contours. Contour labels can be deleted by holding down the SHIFT key while clicking on a label.

The bottom portion of the *Contour Label Options* dialog control how the labels appear. On the left side, the user can control how many digits of accuracy are desired. The default will match the contour legend. On the right side, the user can select a color and font for the label. For labels on contours, the user may also specify that the contour be oriented to lie along the contour.

Contour Legend Options

The Contour Legend Options controls the formatting and location of a displayed legend. If a contour dataset exists and is displayed, the legend will be shown if the Legend check box on the **Contour Display Options** dialog is checked. This window is accessed only from the Legend Options button on the **Contour Display Options** page of the Display Options dialog.

The Formatting section includes fields for the Title and Units displayed with the legend, a Font selection button for text style, and Height and Width fields for legend size.

Since contour datasets can be displayed for multiple modules at the same time and, therefore, multiple contour legends can be displayed, the Title field can include keywords for convenient labeling. The following title keywords are case sensitive:

- "MODULE" – will be replaced with the title of the contour dataset's module
- "DS" – will be replaced with the name of the currently selected contour dataset
- "DS:TS" – will be replaced with the name of the currently selected contour dataset followed by the current time step

A title of "MODULE DS:TS" is best since it will automatically update as contour dataset selection changes.

The Units field includes the single case sensitive keyword of "DEFAULT", which will be replaced with the velocity units of meters per second (m/s) or feet per sceond (ft/s) based on the current coordinate system's horizontal units.

The Location section includes a combo box for specifying the location of the displayed legend. The locations include:

- Top left corner
- Bottom left corner
- Top right corner
- Bottom right corner
- Screen location – specify the location based on screen precentages
- World location – specify the coordinate location

Related Topics

- Color Ramps
- Display Options

New Palette

The New Palette dialog is used to create a new user defined palette. The following palette options can be set:

- Initial Color Ramp Type:
  - Solid Color
  - Intensity Ramp
  - Hue Ramp
  - Elevation
  - Ocean
  - Magnitude Difference
- Color – Only available for **Solid Color** and **Intensity Ramp**
- Number of Colors - Only available for **Solid Color**, **Intensity Ramp**, and **Hue Ramp**
- Palette name

Once the general options have been set in the New Palette dialog, the palette can be fine tuned using the tools in the Color Options dialog.

User Defined Palettes

The user may define and edit a color palette for use with contours. In the Display Options dialog, on the Contours tab, pressing the *Color Ramp* button, opens the Color Options dialog. Inside the Color Options dialog, selecting the User defined option enables the following options:

User Defined Palettes Frame

- New Palette – Create a new color palette. This opens the **New Palette** dialog, which is used to define a color ramp palette. The user selects a preset palette and the initial number of colors in the palette. The palette can be fine tuned once it is created in the Color Options dialog.
- Delete Palette – Delete the selected palette.
- Load Palettes – Load palettes from an SMS defined palette file.
- Save Palettes – Save all of the user created palettes to a file using the format shown above.

Current Palette Frame

The color pallet selected in the **User Defined Palettes** frame is displayed. The user can select, edit, and drag colors in the Color Palette using the following tools:

- Create a breakpoint tool
  - Mouse left click – Creates new breakpoints
- Select an individual breakpoint tool
  - Mouse left click and drag – Changes the value associated with a breakpoint
  - Mouse left double click – Opens the Color dialog to change the color associated with a breakpoint
  - Mouse left click, then DELETE key – Delete the selected breakpoint

Value edit field – Change the value of a selected color. Changing a value will move the color inside the color palette window.

Edit Table – This button opens the Color Table dialog. Values and colors associated with each breakline can be viewed and edited. This dialog is useful for creating a palette with a logarithmic scale. It may be difficult to select colors very close to one another at the lower end of a log scale using the mouse left click button, but the values can easily be specified in this dialog.

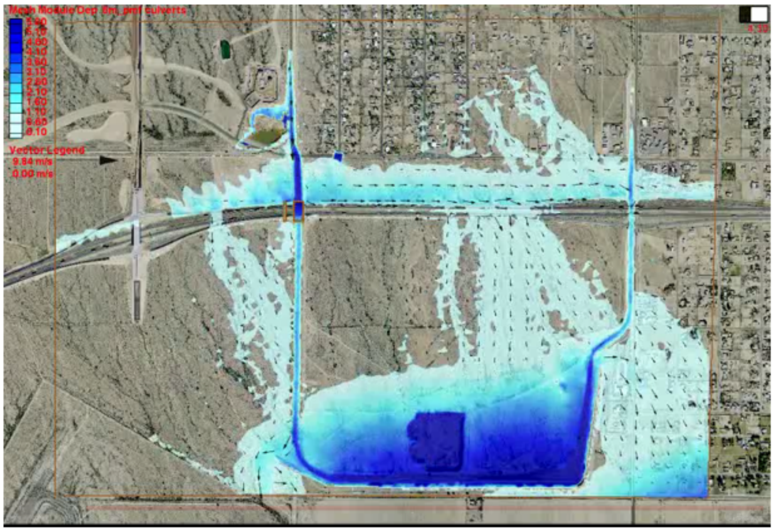
- Display Value As – Show the value of each color as:
  - **Percentages (0.0-1.0)** – A percentage across the palette, with 0.0 being to the left of the palette and 1.0 at the right edge of the palette.
  - **Numerical Values** – The actual value of each color. Each color will represent a value such as elevation.

Related Topics

- Color Palette Files

Vector Display Options

At a glance



Vectors display

- Visualize vector datasets as arrows
- Constant size or vary by magnitude
- Show just a range of magnitudes
- Color by magnitude

SMS can generate contours from a vector dataset. Vector display is enabled using in the Display Options dialog. Unique vector display options can be set for each module that uses vector datasets.

The options used to generate vectors can be edited by:

- The menu command *Data* | *Vector Options*. The data menu is available in most modules. See Module Specific Menus for more information.
- Opening the Display Options dialog.
- The *Vector Options* Macro. 

Vector Display Placement and Filter

Display

The Vector Display Placement and Filter section allows the user to control the density of arrow to be displayed. In a very dense mesh, a large number of data points may be displayed very close together on the screen. Therefore, if a vector is displayed at every point, the picture can become a jumble of vectors on top of each other. One way to avoid this is to zoom in on a specific portion of the mesh, so the nodes are not displayed so close together. However, if the desired region of the mesh is still too dense, or zooming in is not acceptable, the user can filter the displayed vectors. SMS provides the following display locations for filtering vectors:

- **At each node** – Display vectors at each node (data location)
- **At corner nodes only** – Display vectors at corner nodes only (useful for quadratic elements)
- **On a grid** – Display vectors on a grid (uniform grid overlaying the mesh or grid geometry). The x and y spacing of the grid are specified in pixels, so regardless of the zoom level the grid remains constant.
- **On a coverage** – Display vectors on the vertices of feature arcs in the specified coverage (active or specified).

Z-offset

To ensure that the vectors are visible, the Z-offset can be specified to display the vectors a distance above the geometry. The vectors can be filtered further by displaying only a range of magnitudes instead of all vectors.

Arrow Location

Vector arrows can be displayed with the following placement options:

- **Tip** – Display vectors with the vector arrow tip at the data location.
- **Tail** – Display vectors with the vector arrow tail at the data location.
- **Center** – Display vectors with the vector arrow shaft centered over the data location.

Arrow Options

In the Arrow Options section, the user specifies how the arrows will appear in the graphics window. Arrow shaft length can be a constant length, a scaled length, or a range of lengths. The line width of the arrow can also be adjusted. Arrows may be a constant color, or shaded according to magnitude. If a ramp of colors is desired, the color of the vector is extracted from a ramp. By default, the arrow with the smallest magnitude is displayed in the color at the bottom end of the ramp, and the arrow with the largest magnitude is displayed in the color at the top of the ramp. Intermediate magnitudes are interpolated to select an appropriate intermediate color. Alternately, the user can define the magnitudes that map to the top and bottom of the ramp. If this option is used, any arrow with a magnitude lower than the minimum is displayed in the color at the bottom of the ramp, and any arrow with a magnitude greater than the maximum is displayed with the color at the top of the ramp. Users can specify the shape of the arrow head with absolute head length and width values or values proportional arrow length. The style of arrow head is based on the selection of the solid, hollow, and line head types. A preview of the arrows (fixed, or maximum and minimum) based on the selected options are displayed in this section of the dialog.

Legend

The Vector Options also includes a toggle for the display of the vector legend. The vector legend displays the significance of the size of the vectors displayed on the grid. Selecting the legend Options button opens the *Vector Legend Options* window.

Vector Legend Options

The Vector Legend Options controls the formatting and location of a displayed legend. If a vector dataset exists and is displayed, the legend will be shown if the Legend check box on the *Vector Display Options* dialog is checked. This window is accessed only from the Legend Options button on the *Vector Display Options* page of the Display Options dialog.

The Formatting section includes fields for the Title and Units displayed with the legend, a Font selection button for text style, and Height and Width fields for legend size.

Since vector datasets can be displayed for multiple modules at the same time and, therefore, multiple vector legends can be displayed, the Title field can include keywords for convenient labeling. The following title keywords are case sensitive:

- "MODULE" – will be replaced with the title of the vector data set's module
- "DS" – will be replaced with the name of the currently selected vector data set
- "DS:TS" – will be replaced with the name of the currently selected vector data set followed by the current time step

A title of "MODULE DS:TS" is best since it will automatically update as vector data set selection changes.

The Units field includes the single case sensitive keyword of "DEFAULT", which will be replaced with the velocity units of meters per second (m/s) or feet per sceond (ft/s) based on the current coordinate system's horizontal units.

The Location section includes a combo box for specifying the location of the displayed legend. The locations include:

- Top left corner
- Bottom left corner
- Top right corner
- Bottom right corner
- Screen location – specify the location based on screen precentages
- World location – specify the coordinate location

Related Topics

- Contour Options
- Color Ramps
- Display Options
- Visualization

Visualization for 3D Solutions

3D solutions can be viewed on a 3D mesh (VTK mesh module). A 3D solution includes data at multiple z layers and becomes a volume. This is not to be confused with displaying 2D results that represent a surface with 3D coordinates. A 2D solution represents depth averaged values and cannot represent a changing solution in the z direction.

3D Fence Diagram

A 3D fence diagram displays solution data on user specified vertical planes. Fences can be useful to illustrate how a 3D solution varies with depth. Multiple fences can be displayed at the same time to help visualization the solution.

3D fence diagrams allow you to view a cross-section of a 3D solution. To create/view a 3D fence do the following:

Displaying 3D fences requires:

- A 3D mesh with solution datasets.
- A coverage of any type that has one or more feature arcs without any vertices. This defines where the fences will be located. The arcs cannot have vertices since only planar surfaces can be represented.

3D fences can be turned on in the display options dialog. The coverage used for the fence definitions is specified in the display options dialog. The fences will use the current contour settings and are always represented with color-filled contours.

Remember to rotate out of plan view to see the fence.

Iso-surfaces

Iso-surfaces can be used to display 3D solutions. The display options for iso-surfaces are set using the contour options in the Display Options dialog.

2.7. Export Options

Export Tabular File

SMS can export much of the data managed and displayed int the system to a tabular data format. This type of file is sometimes referred to as a "*.csv" or comma separated values file. In actuality, the delimiter may be commas, spaces, tabs, or other typical white space characters.

The data to be exported depends on the active module when the command is issued. For example, when the Mesh module is active, mesh nodes will be saved and when the Scatter module is active, scatter vertices will be saved. The user supplies the file name to contain the data.

If data points are selected when the command is issued, the option is given to output all data points of the defined type, or only the selected points. A default header is provided which defines the number of data points represented.

The user then selects the nubmer of columns and number of digits of precision to save each value. For each column the user then selects the data to be stored in that column. This can include the *x location*, the *y location* or any data set currently loaded into SMS for that data type. Transient datasets may be saved in a range of columns.

Exporting Profile Dialog

The Exporting Profile dialog allows you to export the plot data.

Export

- Image Export
  - EMF
  - WMF
  - BMP
  - PNG
- Text / Data

Export Destination

- ClipBoard
- File – Creates an ASCII text file of the data
- Printer – Exports the data to your printer

Export Size

If exporting to an image format, allows the image size and resolution to be specified.

Related Topics

- Plot Window

Export Meshes in MIF/MID Format

Export

To export maps in MIF/MID format:

- Load or create an existing map
- Select "File→Save As"
- Select "Save as type:" and choose"Catalog file (*.xml)"

An XML file is created which outlines the directories of the MIF/MID files. A points.mif/mid, polygons.mif/mid and polylines.mif/mid (if exists) is also created.

Import

Open the saved XML file to import the mesh from a MIF/MID file.

Export Dataset Dialog

The *Export Dataset* dialog is used to export scalar or vector data sets. To open the *Export Dataset* dialog, use the dataset right click menu.

- File Type
  - Binary Dataset Files (*.dat) – Benefits include fast read/write times, small file size.
  - ASCII Dataset Files (*.dat) – Can be imported into Microsoft Excel and viewed with standard text editors.
  - HDF5 Dataset Files (*.h5) – Benefits include fast read/write times, small file size, native compression.
- Time Steps
  - Current time step – Exported dataset will only contain the current time step.
  - All time steps – Exported dataset will contain all time steps.
- Filename – Path and filename used for exported data set.

Related Topics

- Binary Dataset Files (*.dat)
- ASCII Dataset Files (*.dat)
- XMDF Files

Export Map Data in Shape Format

Export

To export maps in shape format:

- Load or create an existing map
- Select "File–Save As"
- Select "Save as type:" and choose "Catalog file (*.xml)"
- In the menu item, select "Shape" as the format and verify the correct coverage is select (has a check mark)
- Click "Ok"

An xml file is created which outlines the directories of the shape files.

If the coverage created is a generic 2dm, feature point and feature arc boundary condition data will also be saved.

Import

Open the saved *.xml file to import the data from the shape file.

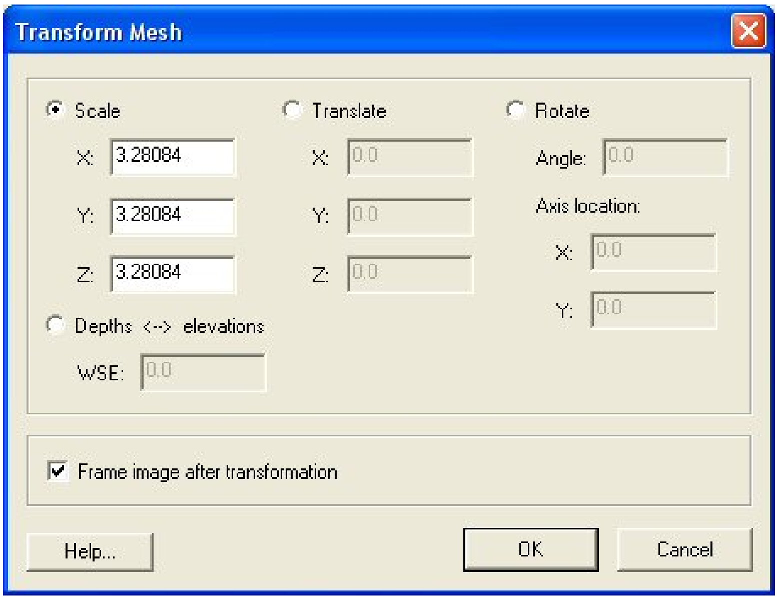
2.8. Geometric Tools

Data transform

At a glance

- Data can be scaled, translated, rotated
- Depths/Elevations can be converted back and forth

The Transform command is used to move scatter points. The user is asked which will be transformed, the active set or all sets. In the dialog that appears, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available:



Transform Data

- **Scaling**: Scaling factors for the X, Y, and/or Z directions are entered. To prevent scaling a specific direction, the default value of 1.0 should be used.
- **Translation**: Translation values for the X, Y, and/or Z directions are entered. To prevent tranlation in a specific direction, the default value of 0.0 should be used.
- **Rotations**: When rotation is selected, the set of options on the right side of the dialog become available to define the center of rotation. If the *Specified Point* option is used, then the center of rotation is explicitly defined. Otherwise, after clicking the OK button from the *Nodes Transform* dialog, the user must click in the graphics window at the Point or on the Node about which the rotation should occur. The rotation will occur counter-clockwise by the specified angle around the specified center of rotation.
- **Datum Conversions**: Convert between elevation and depth data.

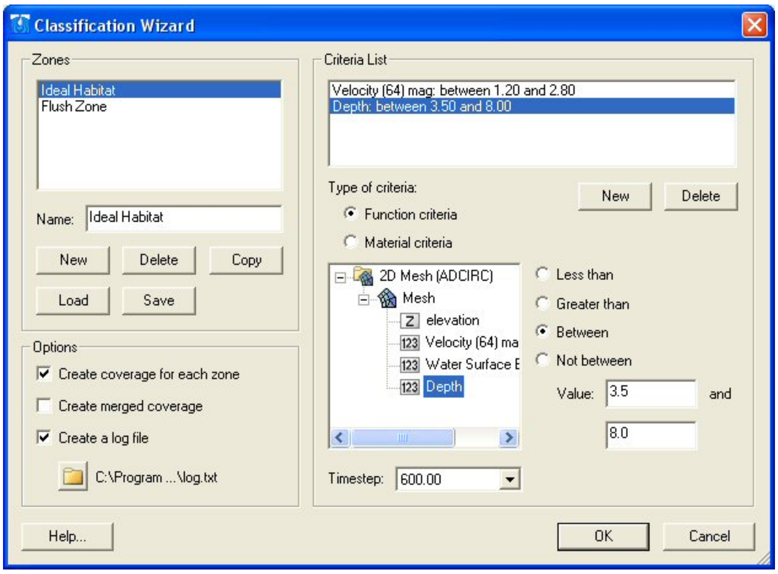
By default, the image will be framed after the transformation takes place. However, this can be turned off by using the *Frame Image After Transformation* option.

Related Topics

- Scatter Data Menu

Zonal Classification

At a glance



Zonal Classification

- Generate a map coverage identifying areas that meet specific requirements
- Requirements can be based upon dataset values such as less than a specific value or based upon materials in an area property coverage

Zonal classification is a tool that will identify areas that meet a set of criteria. The criteria can be based upon scalar dataset values and/or specific material ids in a coverage.

A zone may contain one or more criteria. A zone may identify areas that have a range of depths and also a range of velocities. Multiple zones can be evaluated at the same time. If you are using multiple zones, you can have SMS create a separate coverage for each zone, a coverage that includes all the zones where each polygon's material identifies the zone or zones the polygon is valid for, or both a coverage for each zone and a merged coverage.

Zones and criteria associated with them can be saved and loaded from within SMS. This makes it easier to evaluate multiple scenarios using the same set of criteria.

If desired, SMS can create a log file that contains information such as the areas found in each zone.

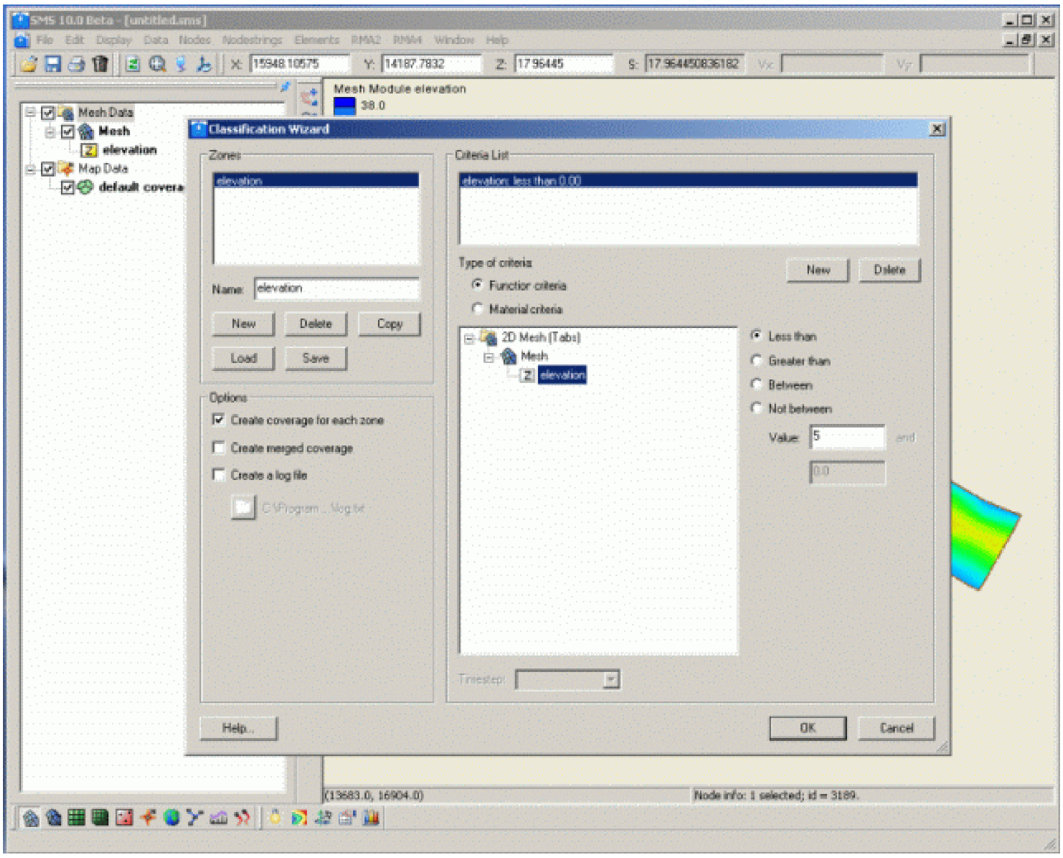
Example

One application of zonal classification is to help quantify the amount and quality of fish habitat. Certain types of fish prefer or require different depths, velocities, and substrate. These preferences depend upon the life-cycle stage for fish.

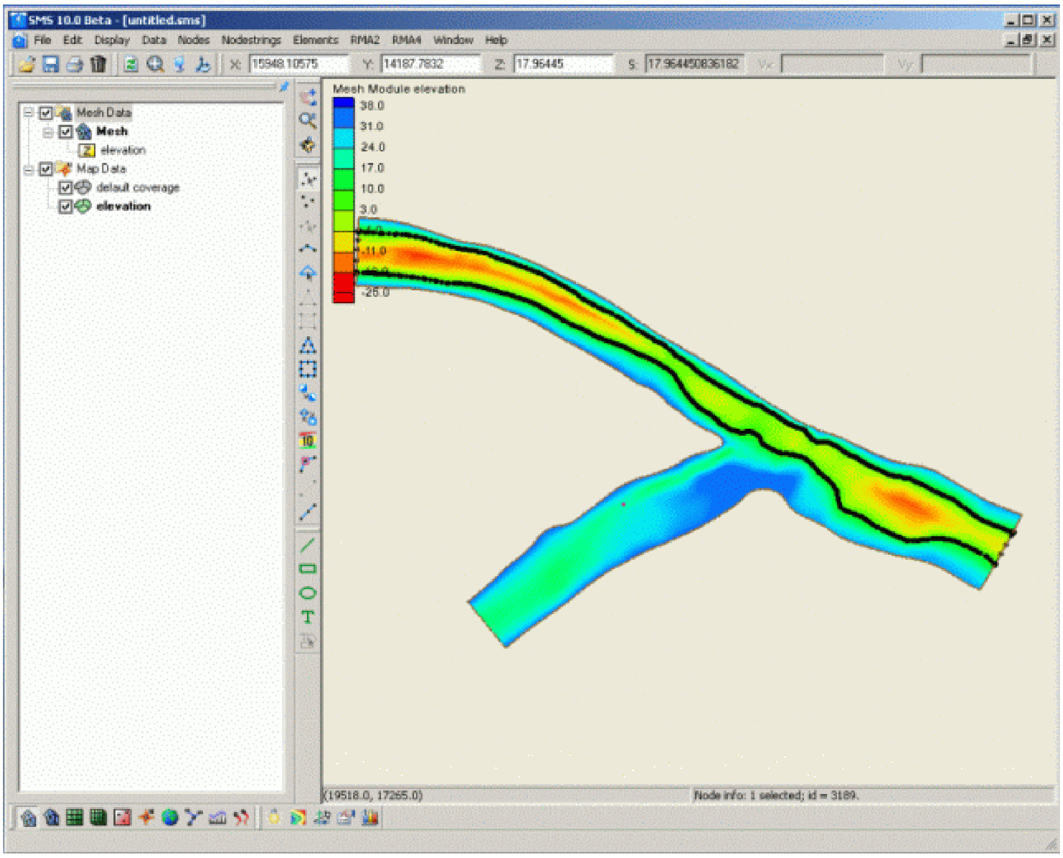
The following demonstrates how you could use zonal classification for a very simplified example to identify areas meeting a certain set of criteria. The example is fictitious and uses made up criteria.

For the first sample criteria we will identify areas that have an elevation of 5 ft or less. This is done by:

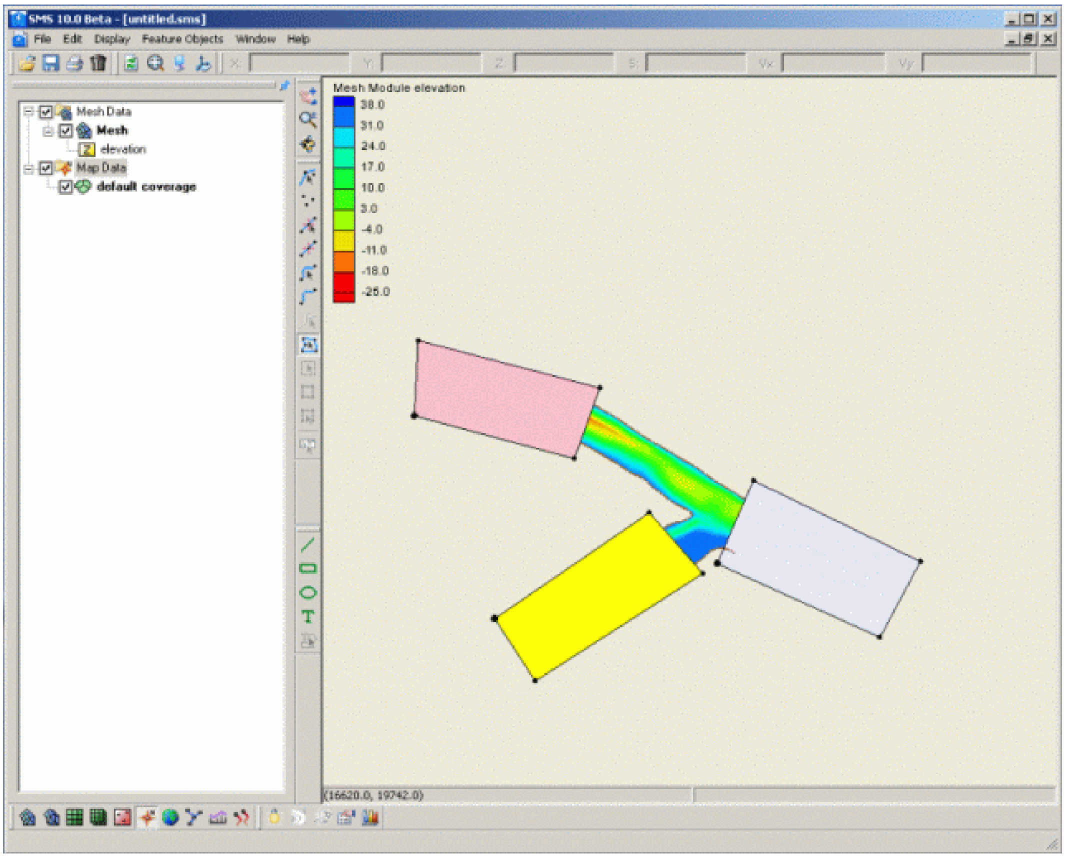
1. Creating a new zone based upon this criteria.
2. Creating a new criteria based upon a functional criteria.
3. Specify the elevation dataset and the criteria to be less than 5 ft.



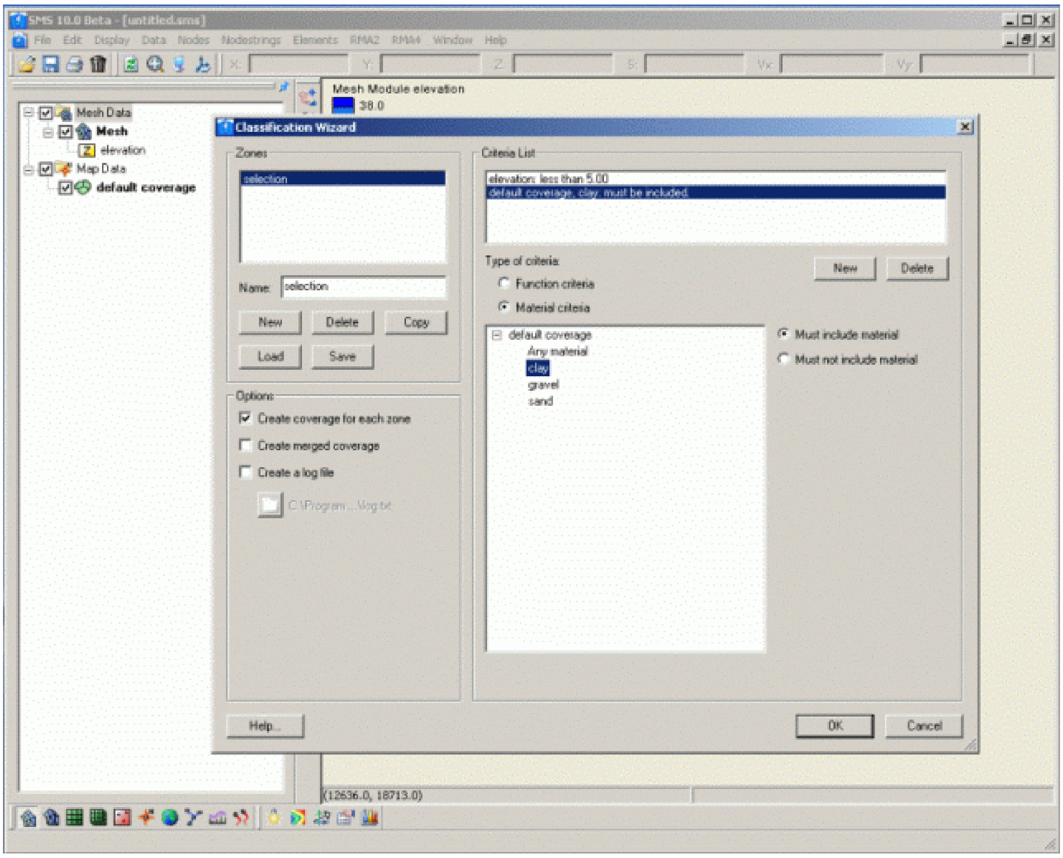
After executing the zonal classification a new coverage was created and polygons identify the areas that meet the requirements of the zone (in our case the elevation less than 5 ft). Assuming that our criteria identifies a target habitat, it is easy to see the areas that meet the criteria.



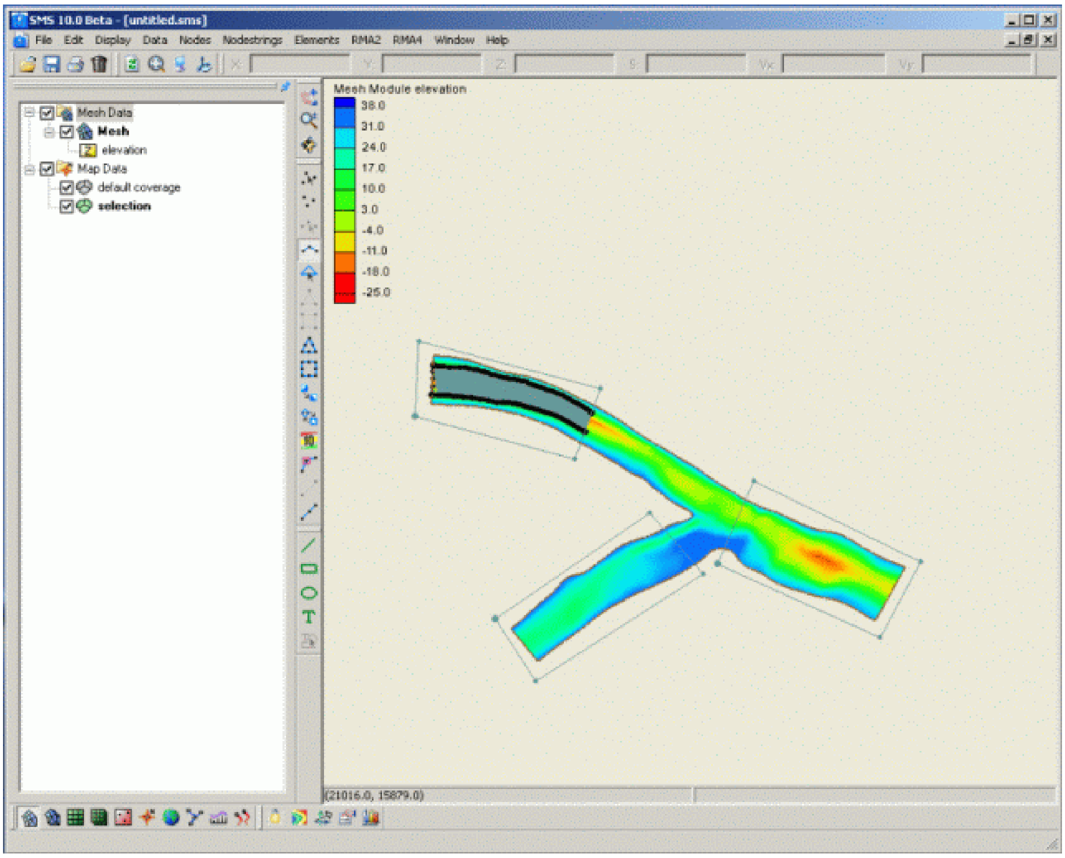
In addition to the elevation, let's assume that our target habitat also requires a specific type of substrate (bottom sediment type). For this example, we have created polygons in a area property coverage and identified areas with different substrates (again this information is fictional).



We added another criteria to our original zone so that the zone only includes areas that have clay substrate.



Now polygons only exist where the elevation is less than 5 ft and the substrate is clay.



External Links

- Vector based classification of zones from distributed datasets or GIS polygon data – Russell D. Jones, 2003 – Thesis (M.S.) Brigham Young University Dept. of Civil and Environmental Engineering. TA 4.02.J65482 2003

2.9. Images

Images

At a glance

A color aerial photo under a semi-transparent topographic map

- Multiple images can be read/viewed at the same time
- Many image formats are supported including JPG, TIFF, PNG, MrSID, and ECW
- Images can be geo-referenced to view images along with other data
- Images can be draped over mesh or scatter data
- Image pyramids can be created which improves viewing at various zoom levels

A background image is a digital picture detailing topographic and land use attributes of an area of interest. In SMS, these digital pictures are typically maps or aerial photos that are useful in locating and defining the boundaries of the study area and the extents and features in the project domain. Images can be imported to SMS and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance a plot. Images can also be draped or "texture mapped" or draped onto a scatter dataset (TIN) or finite element mesh.

Supported Image File Formats

- Enhanced Compression Wavelet [1] (*.ecw)
- Graphics Interchange Format [2] (*.gif)
- Joint Photographic Experts Group [3] – (*.jpg/jpeg)
- Multiresolution Seamless Image Database [4] – (*.MrSID)
- Tagged Image File Format [5] – (*.tiff)

Importing an Image

Images can be opened in SMS using the *File* | *Open* menu command. They can also be added to a simulation by dragging and dropping the file into SMS. The images are then added to the image folder in the Project Explorer and displayed in the background to aid in the placement of objects as they are being constructed or simply to enhance visualization of the project domain. All TIFF images are converted to JPEG when they are read in. Multiple images can be imported into SMS.

Exporting Image Files

Images (or files related to images) are saved in the following ways:

Save As

The image displayed in the Graphics Window can be saved as a Bitmap Image File (*.bmp) or JPEG Image File (*.jpg, *.jpeg) using the *File | Save As* menu command and specifying an image file as the save as type. The resolution of the saved image is based on the screen resolution and scale factor specified in the Preferences dialog.

Project File

When a project file is saved any images that are part of the project are saved. The registration information is saved in the project file to provide the coordinate system information for the image.

Copy to Clipboard

When the *Edit | Copy to Clipboard* menu command is selected, the image currently displayed in the Graphics Window is copied to the clipboard. This image can then be pasted into reports or other programs by pressing *CTRL + V*. The resolution of the saved image is based on the screen resolution and scale factor specified in the Preferences dialog.

Export World File

A World File can be exported for the selected image by right-clicking on the Project Explorer and selecting the *Export World File* command. A world file is a special file that contains registration data that can be used to register images.

Geo-Referencing

A geo-referenced image includes information specifying the real world size and location of the image. The coordinate system can be embedded in the file or given in a separate file called a world file (for example: a tiff world file, *.tfw). When geo-referenced image files are opened, SMS automatically registers the image to the real world coordinate location specified. In the case where a separate world file is used, SMS will search for it and register the image if the world file has the same filename prefix as the image file and is in the same folder.

If the image file is not geo-referenced then you will have to register the image manually. (See Registering an Image)

When the SMS project is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Display Options

Image display options are changed in the Project Explorer. Display options include:

- **Visibility** – The visibility of an image is turned off by toggling the check box next to the image in the Project Explorer.
- **Transparency** – The transparency of each image can be changed by right-clicking on the image in the Project Explorer and selecting *Transparency* from the right click menu.

Image Deletion

A single image is deleted by right-clicking on the image in the Project Explorer and selecting the *Delete* command. To delete all images, you right-click on the Images folder in the Project Explorer and select *Clear Images*.

Dynamic Imagery from ArcGIS

Starting in SMS 11.0 (32-bit only), dynamic background images can be accessed from the web through ArcGIS. If you have ArcGIS on your computer, you can use the GIS module within SMS to get background imagery that updates on the fly from the internet. To access these images, follow the steps below.

1. Switch to the GIS module (select the globe in the bottom left of the SMS screen)
2. Select “Data | Enable ArcObjects”
3. Select “Data | Add Data…”
4. Browse to the “C:\Program Files (x86)\SMS 11.0\Supporting Files\GIS Layer Files” directory
5. Select the desired layer
6. Select “Add”

Note: This feature is only available in the SMS 11.0 beta (32-bit). Since this version is still in beta, it should not be used in place of the release version of SMS (SMS 10.1).

Related Topics

- Image Pyramids
- Import from Web
- Registering an Image

References

[1] http://en.wikipedia.org/wiki/Enhanced_Compressed_Wavelet_File
[2] http://en.wikipedia.org/wiki/Gif
[3] http://en.wikipedia.org/wiki/Jpeg
[4] http://en.wikipedia.org/wiki/Mrsid
[5] http://en.wikipedia.org/wiki/Tiff

Image Pyramids

When an image is imported, the user has the option to “Build Pyramids.” The build pyramids option enhances the on-screen display of images by resampling the imported image at different resolutions. Based on the current extents of the graphics window, one of the resampled images is displayed to deliver the best on-screen display.

There are a few points to keep in mind when building pyramids. The initial generation of pyramid files can take several minutes, depending on the size of the original image and your computer hardware. Building pyramids uses more memory RAM, but the software performance is not otherwise affected. Building pyramids may not improve the on screen display of all images.

Image Preferences (SMS Only)

When pyramids are built for an image, up to four JPEG images are saved to disk. These image files can be saved to a temporary folder or can be saved in the same directory as the original image so that they are not regenerated each time the image is loaded. The user’s preferences for building pyramids can be set in the images tab of the preferences dialog (*Edit* | *Preferences*).

Related Topics

- Import from Web
- Registering an Image

Import from Web

Overview

The *Import from Web* feature allows you to connect to the internet to download free data – images, elevation data etc. If you have an internet connection, this is an easy and convenient way to acquire this type of data.

The data is made available for free by various entities who provide web services [1]. Each XMS programs has a number of available data types they can retrieve.

The *Import from Web* feature is no longer is use for GMS as of GMS 9.0. The feature is still used in GMS 8.3 and earlier.

GMS	SMS	WMS
• NED data – USGS [2] • ASTER and SRTM data – USGS & NASA [4] • TerraServer aerial photo [3] • TerraServer urban [3] • TerraServer topo [3]	• TerraServer aerial photo [3] • TerraServer urban [3] • TerraServer topo [3]	• NED data – USGS [2] • ASTER and SRTM data – USGS & NASA [4] • NLCD and CORINE (European) Land Cover data [5] • World Imagery [6] • World Street Maps [7] • World Topo Maps [8] • MapQuest OpenStreetMap Worldwide Street Maps [9] • Land Use Shapefiles [10] • STATSGO and SSURGO Soil Type Shapefiles [11] • Harmonized World Soil Database v 1.1 [12]

		• Global Land Cover [13]

Data Availability

Elevation (NED, ASTER, and SRTM) Data

- NED data contains the best available raster elevation data of the conterminous United States, Alaska, Hawaii, and territorial islands. NED data are not available for other areas.
- ASTER and SRTM data are available for most of the earth's surface. The ASTER dataset is reliable and high-quality.

Imagery

All the imagery (World Imagery, Street Maps, Topo Maps, and OpenStreetMap.org data) are available for anywhere on the earth. Besides downloading these images using the Import from Web command, these images can be read as online maps and then converted to a static map (stored locally on your computer) for faster display after you have decided on a project area.

Land Cover Data

- The 100 m Resolution CORINE dataset (raster) is available for anywhere in Europe.
- The 30 m NLCD dataset (raster) is only available for the conterminous United States.
- The Land Use Shapefile dataset is currently only available for the states of Utah and Louisiana.
- The Global Land Cover dataset is currently only available for Morocco. If you are interested in making data available for your region, you can either follow the following steps or contact Aquaveo, who can make data for your area available on their web servers for a fee. Follow the following steps to bring Land Use data for your area into WMS:

1. Go to the European Space Agency site to download land use data [13].
2. Download the .zip file Globcover2009_V2.3_Global_.zip and unzip this file on your computer.
3. Open GLOBCOVER_L4_200901_200912_V2.3.tif in a GIS (such as ArcMap) and convert it to an ESRI raster file. Trim the raster as needed, then convert the raster to a shapefile.
4. Convert the file Globcover2009_Legend.xls to a *.dbf file and join this file with the shapefile values to get the land use names and IDs.

Soil Data

- SSURGO soil datasets are available for some parts of Utah, Colorado, and California and are available for most of Louisiana.
- A STATSGO soil dataset is available for all of Utah. You can also download the STATSGO dataset for the entire United States, but this is a large download (338 MB) and will take some time to download.
- Data from the Harmonized World Soil Database are only available for Morocco. If you are interested in making data available for your region, you can either follow the following steps or contact Aquaveo, who can make data for your area available on their web servers for a fee. Follow the following steps to bring soil data for your area into WMS:

1. Download and install the Harmonized World Soil Database program to your computer from the Harmonized World Soil Database web site [14].
2. Launch the HWSD Viewer on your computer. The soil data will be copied to the folder c:\program files (x86)\HWSD_v<xxx>\Data where <xxx> is the version of the viewer you have downloaded. The program may

also be installed in c:\program files\<...> if you are running a 32-bit version of Windows. The following files are contained in this folder:

   1. The HWSD Raster *.zip file.
   2. The HWSD DBF file.
   3. The HWSD_META DBF file.

3. Copy the files in the data folder to a writable location on your computer and unzip the HWSD Raster *.zip file.
4. Open the .bil file in ArcMap and convert the *.bil file to a shapefile using the IDs.
5. Join the HWSD DBF file with the IDs in the shapefile.
6. Join the attribute IDs with the HWSD_META DBF file. This gives you a shapefile with the soil IDs and various soil attributes that can be used for hydrologic modeling in WMS.

Additional Information

Note that more vector-based soil and land use datasets are available; Contact Aquaveo [15] if you are interested in adding data from a specific area to the list of available land use or soil data that can be downloaded. A comprehensive list of soil and land use data available for download is located here.

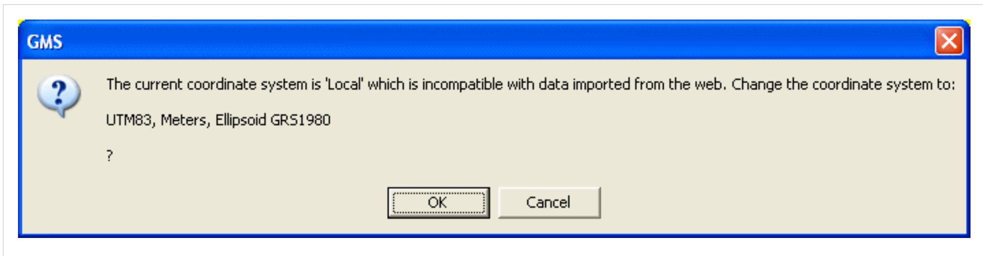
Terraserver images are no longer available because this web service has gone offline.

Using the Import from Web Command

This feature is found via the ***File|Import from Web*** command. This command results in the following series of dialogs.

1. **Warning**

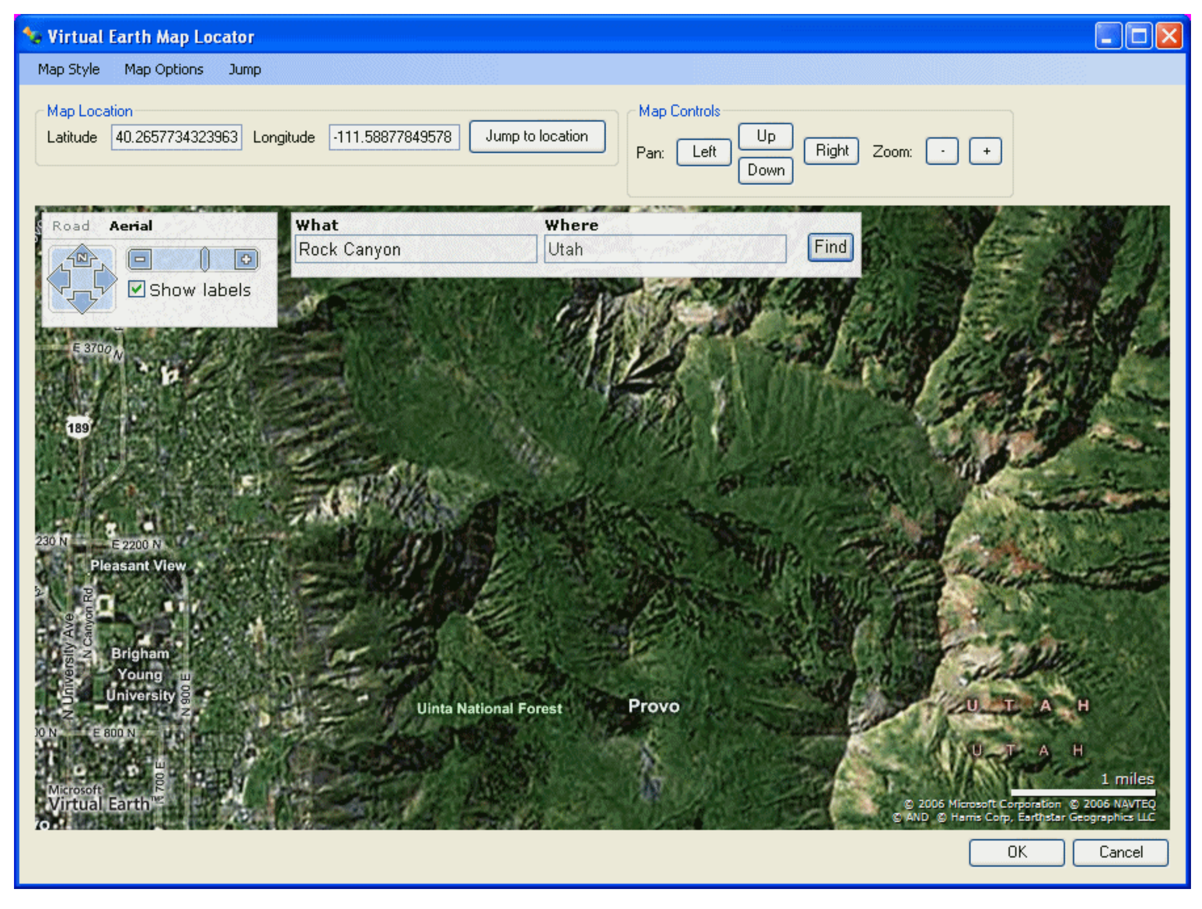
   If your current coordinate system is "Local", you will get the following warning dialog. The data that will be downloaded is georeferenced so your project coordinate system must be something other than 'Local'. Selecting **OK** will change the current coordinate system.



2. **Virtual Earth Map Locator**

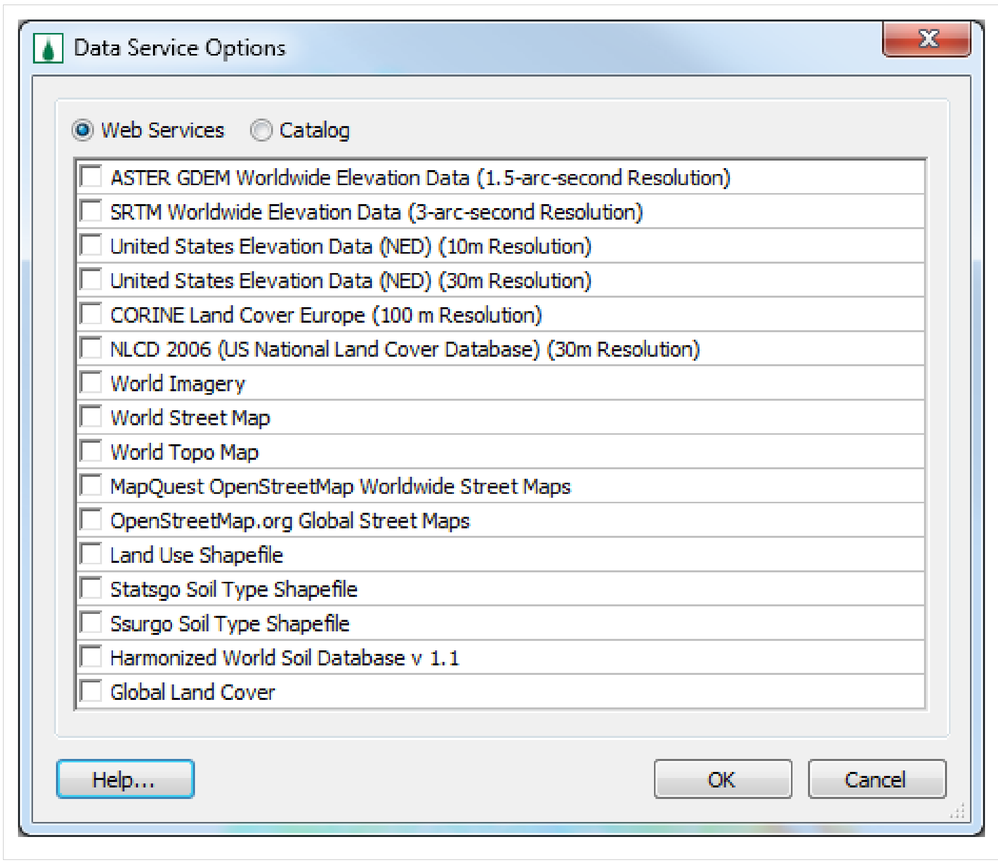
   Use the map in this dialog to go to your location of interest.

- You can **zoom** in or out using the controls or the mouse wheel.
- You can **pan** using the controls or by clicking and dragging. You can also enter the latitude and longitude to jump to a specific location.
- Use the *Map Options* menu to turn on the floating controls in the map (search, pan and zoom).
- Use the *Map Style* menu, or the floating controls, to change the map between **Road**, **Aerial**, and **Hybrid**.



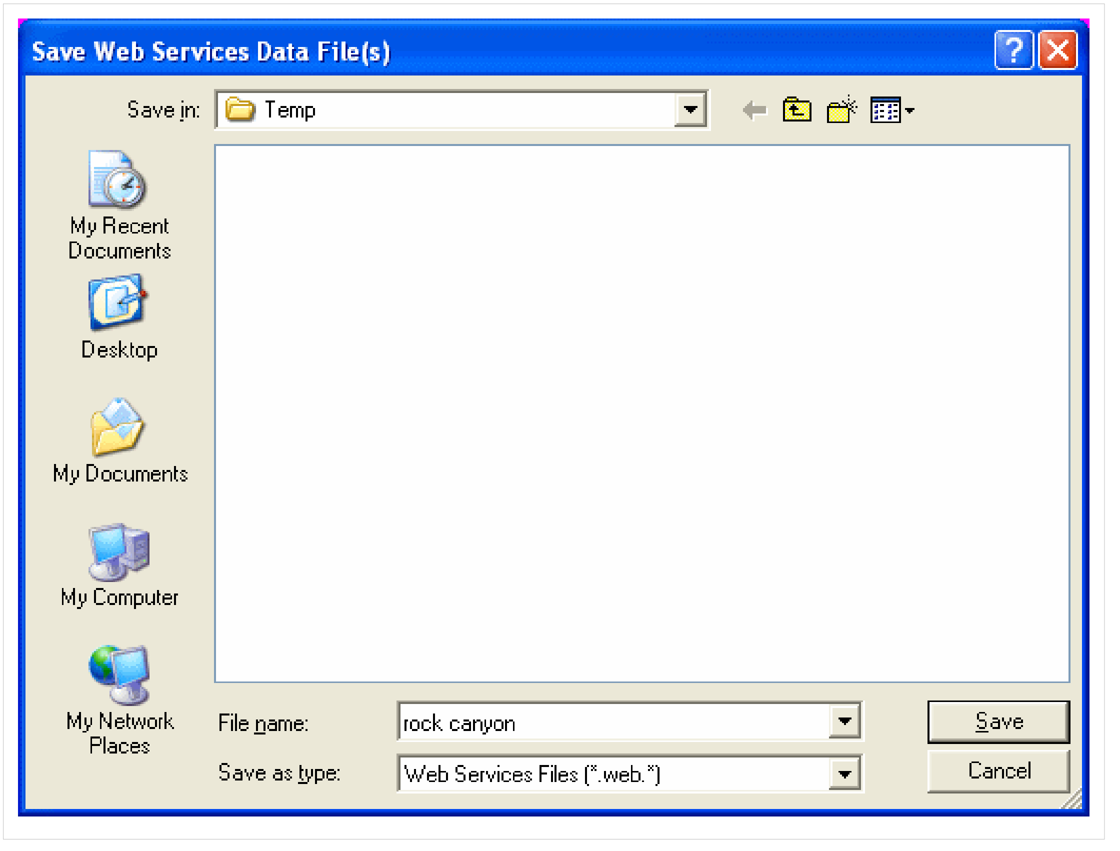
3. **Data Service Options**

   Here you select which type of data you are interested in.



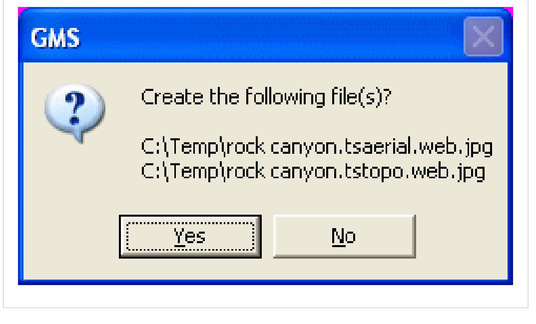
4. **Save**

   Now you are asked where you want to save the data. You only need to specify one file name, even if you've selected more than one type of data in the previous dialog. The files will all be given the same prefix but different suffixes.



5. **Confirm File Creation**

   You may be asked to confirm that you want to create the files.

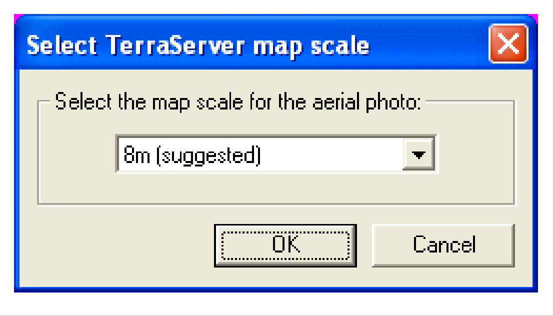


6. **Initialize Connection**

   The following dialog is shown while the connection is being made.

7. **Select Scale**

   Smaller numbers (larger scales) will result in better resolution, but longer download times.



8. **Downloading**

   This dialog reports the download progress. If you click **Abort**, your image will exist but will be only that portion that you've downloaded so far.

Steps 6-8 will repeat for each data type you selected in step 4. After everything is finished, the data (images etc.) will appear in the Project Explorer.

References

[1] http://en.wikipedia.org/wiki/Web_service
[2] http://ned.usgs.gov/
[3] http://www.terraserver.com/
[4] http://srtm.usgs.gov/
[5] http://seamless.usgs.gov/nlcd.php
[6] http://services.arcgisonline.com/ArcGIS/rest/services/World_Imagery/MapServer
[7] http://services.arcgisonline.com/ArcGIS/rest/services/World_Street_Map/MapServer
[8] http://services.arcgisonline.com/ArcGIS/rest/services/World_Topo_Map/MapServer
[9] http://developer.mapquest.com/web/products/open
[10] http://waterdata.usgs.gov/nwis
[11] http://soildatamart.nrcs.usda.gov/
[12] http://www.iiasa.ac.at/Research/LUC/External-World-soil-database/HTML/
[13] http://ionia1.esrin.esa.int/
[14] http://webarchive.iiasa.ac.at/Research/LUC/External-World-soil-database/HTML/
[15] http://www.aquaveo.com/contact-us

Registering an Image

If an image file is not geo-referenced then the user must define the coordinate system of the image. The register dialog allows the user to specify the coordinate system for the image. When an image is opened, if the image is not self-referenced, XMS attempts to find world file with the same name as the image (*.wld or *.jpgw extension). If neither of these is found, the register dialog opens.

What is Image Registration?

Before an image can be displayed, the image must be "registered" or geo-referenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window. If an image is not registered properly, any objects which are created using the background image as a guide will have the wrong coordinates.

Register Image Dialog

An image is registered using the Register Image dialog. The main feature of the Register Image dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- **2 point or 3 point registration** – Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- **Import World File** – Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.
- **Image name** – Used to associate a name with the file. This name will appear in the project explorer.

Register Image Dialog Tools

The following tools can be used to help position the registration points:

Tool	Tool Name	Description
	Select Point Tool	The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.
	Zoom Tool	In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom Tool is used to zoom in a portion of the image.
	Pan Tool	After zooming in on a portion of the image, the Pan Tool is used to pan the image vertically or horizontally.
	Frame Macro	The Frame Macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.

Import World File

The **Import World File** button can be used to automatically define the registration data. A world file is a special file associated with a previously registered image that is exported from ArcView® [1] or Arc/Info® [2]. The file contains registration data that can be used to register the image.

Saving/Reading Image Registration Data

When a project file is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Convert Point Coordinate System

The x, y coordinates of each register point must be specified. If the user has the (x,y) coordinates in a different coordinate system than their project, the coordinates will need to be converted.

GMS Point Conversion

The Convert Point button in the image registration dialog will allow the user to convert the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

An alternative approach is to convert the coordinate system after importing by right clicking on the image in the Project Explorer and choosing ***Coordinate Conversion*** from the right click menu.

WMS Point Conversion

The Single Point Conversion command in the **Edit** menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

References

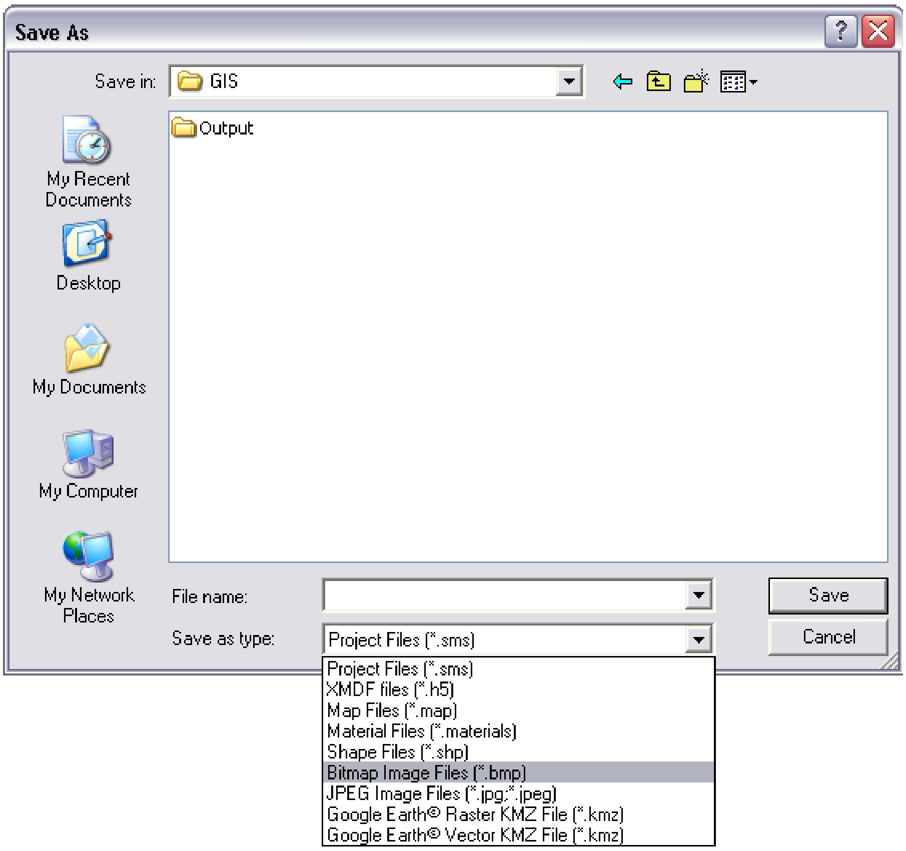
[1] http://www.esri.com/software/arcview/
[2] http://www.esri.com/software/arcgis/arcinfo/

Save as Image

Saving in image format

It is now possible to save information displayed in the graphic windows in image format. Images can be saved in the following two formats:

- Bitmap Image Files (*.bmp)
- JPEG Image Files (*.jpg or *.jpeg)



It can be useful to save the contents displayed in the graphics window so the images can be used for presentation purposes, documents, etc.

Web Service for Background Imagery

SMS supports the ability to obtain image data from web servers. The imagery will update as you pan and zoom and use an appropriate resolution for your current zoom level. Since you are obtaining the information over the internet, the performance of these images will be dependent upon the speed of your internet connection.

System Requirements

SMS 11.0 and ArcGIS 9.3 or above are needed for this feature.

Projection

To use web layers, you must be in a non-local projection.

GIS Web Layers

SMS ships with several layers that you can use. Some of the layers are specific to the US and others are worldwide. You may want to experiment with different layers as some will give better performance or quality. To find the files shipped with SMS, go to the windows menu, go to the SMS 11.0 folder, and click on the item labeled "supporting files." This will open an explorer window to the folder that contains supporting files used with SMS. You will see a folder named "GIS Layer Files." This folder contains the GIS web layers that ship with SMS.

In order to load the GIS layers, you have to be using the ArcObjects interface inside of SMS. This is activated by switching to the GIS module and selecting the menu item, Data | Enable ArcObjects.

Zooming in to view more details

As you zoom in, more of the GIS layer features such as roads, peaks, etc will be more visible (as well as their labels in some cases depending on the GIS layer opened). The more you zoom in, the more details will appear. When you zoom out, the details will become less visible.

Back to XMS

2.10. Preferences

Preferences

The ***Edit | Preferences*** command brings up the Preferences dialog. The Preferences Dialog contains the following tabs:

General

- **File IO**
  - XMDF File Compression – Use compression when saving XMDF files.
  - Override temp directory – Specify the location where SMS temporary files are written.
- **View Data File Option**
  - Ask for Program – SMS will prompt the user to specify the program used to open a data file when the *File | View Data File* command is called.
- **Deletions**
  - Confirm Deletions – Whenever a set of selected objects is about to be deleted, the user can choose to be prompted to confirm the deletion. This is meant to prevent accidental deletion of objects.
- **Model Priority** – Models can be launched using a particular process priority. This priority specifies how the operating system should treat the process. We recommend using the "Above Normal", "Normal", or "Below Normal" options in most circumstances. The options are as follows:
  - Realtime – Highest priority. May cause machine to become unresponsive. Use with extreme care.
  - High – Only allows realtime process to go before it. Can use nearly all CPU cycles. Use with extreme care.
  - Above Normal – Takes priority over normal processes. Will take CPU cycles before normal applications do.
  - Normal – No special scheduling takes place. This is the normal default.
  - Below Normal – Allows processes with normal priority to run first, but runs before low priority processes.
  - Low – The process will only run when the system is idle.
  - Default – The process will be launched with the same priority as it's parent (SMS in this case).
- **Copy to Clipboard**
  - Scale factor – When copying the contents of the main graphics window to clipboard, the resolution can be increase by specifying a scale factor greater than 1.0.

Startup

- Default Module – Specify the active module at startup.
- Default 2D Mesh Model – Specify the active 2D Mesh Module Model at startup.
- Default 2D Cartesian Grid Model – Specify the active 2D Cartesian Grid Module] Model at startup.
- Default Coverage Type – Specify the active Map Module Coverage Type at startup.
- Check version on startup
- Default TUFLOW Executable – Specify the executable to use by default when creating a new TUFLOW simulation. The options are double and single precision for both 32 and 64bit.

Images

- Image Pyramids – Specify whether SMS will always, never, or prompt to build image pyramids.
- TIFF → JPEG Conversion – When a TIFF file is opened in SMS, the file is copied and saved as a JPEG file. You can specify where the JPEG copy of the image file is saved.

File Locations

- Model Executables – Specify the location of numerical model executables.
- Other Files – Specify the location of files used by SMS (LeProvost tidal database, LATLON conversion files, etc.).

Project Explorer

Options to:

- Force active scalar and vector datasets to be in the same folder
- Add diagnostic files when reading model solutions

Toolbars

Options to turn the following toolbars on or off:

- File Toolbar
- Module Toolbar
- Display Toolbar
- Optional Macro

Time

See the Time Step Window article for an explanation of absolute and relative time. The default format of the time steps in the Time Step Window can be set.

Available times options

This option controls which times are displayed in the time step window. The available options are:

- Active datasets (current module only) – The times displayed in the timestep window are based only upon the active scalar and vector datasets in the current module. If neither of these datasets is transient, the time step window will not be displayed.
- All available times (all modules) – The times displayed in the timestep window are based upon times used by any transient object in SMS (includes datasets, some kinds of coverages, and PTM particle sets). All of the times from each of the objects will be used regardless of whether or not the object is active or visible.

Dataset time step rounding

The dataset being used for contours, vectors, or other display option may not have a timestep that corresponds exactly with the time currently chosen in the time step window. When this happens, SMS has two options for determining the values used by the dataset. These options are:

- Interpolate to exact time – Interpolate the dataset values for the selected time step from the nearest time steps before or after the display time. If the display time is before/after all of the time steps the nearest time step is used.
- Use nearest time – The dataset time step nearest the display time will be used (no interpolation).

Map

Options to:

- Snap feature objects to displayed inactive coverage nodes and vertices when creating new feature objects.

Graphics

- **Active Graphics Library**
- **Options**
- Automatically refresh after an edge swap
- Use vertex buffer objects (VBOs) – This option can be specified to change how SMS works with the graphics card. Vertex buffer objects are generally faster and often uses less of the computers main memory. However, there are circumstances where using vertex buffer objects can be significantly slower, such as when the machine's graphics card only has a small amount of onboard memory. By default, this option is on.

Related Topics

- Edit Menu
- Time Settings

Images

- Image Pyramids – Specify whether SMS will always, never, or prompt to build image pyramids.
- TIFF → JPEG Conversion – When a TIFF file is opened in SMS, the file is copied and saved as a JPEG file. You can specify where the JPEG copy of the image file is saved.

File Locations

- Model Executables – Specify the location of numerical model executables.
- Other Files – Specify the location of files used by SMS (LeProvost tidal database, LATLON conversion files, etc.).

Project Explorer

Options to:

- Force active scalar and vector datasets to be in the same folder
- Add diagnostic files when reading model solutions

Toolbars

Options to turn the following toolbars on or off:

- File Toolbar
- Module Toolbar
- Display Toolbar
- Optional Macro

Time

See the Time Step Window article for an explanation of absolute and relative time.

The default format of the time steps in the Time Step Window can be set.

The following Time Step Window options:

- Available times based on:
  - Active datasets (current module only)
  - All available times (all modules)
- Dataset time step rounding – When times are shown from multiple datasets, the time selected in the *Time Step Window* may not be a time step used by all datasets. SMS can:
  - Interpolate to exact time – interpolate the dataset values for the selected time step from the nearest time steps that do have data. If the selected time step is before/after all of the time steps, the first/last time step values will be used.
  - Use nearest time – use the dataset values from the nearest time step.

Map

Options to:

- Snap feature objects to displayed inactive coverage nodes and vertices when creating new feature objects.

Related Topics

- Edit Menu

Time Settings

Transient dataset time values are displayed in the Time Step Window using either a relative time format (e.g., 100.0) or an absolute date/time format (e.g., 1/12/2006 3:23:48).

Zero Time

The zero time represents the date/time corresponding to time t=0. If a dataset does not have an assigned reference time, it will use the global zero time as its reference time.

Absolute Date/Time

When the display format is set to *Absolute Date/Time*, a date/time is shown in the Time Step Window. The date and time format can also be specified.

Relative Time

When the display format is set to *Relative Time*, the days, hours, minutes, and seconds from the dataset reference time is shown in the Time Step Window. The display format for days, hours, minutes, and seconds can be specified. If a decimal format is chosen, the precision can also be specified.

Changing Time Settings

To change the time settings, select the *Menu* command Edit | Time Settings or right click on the time step window in the Project Explorer and select "Time Settings" to open the *Time Setting* dialog.

Related Topics

- Layout of the Graphical Interface

2.11. Spectral Energy

Spectral Energy

The Spectral Energy dialog can be opened through *STWAVE | Spectral Energy*, *BOUSS-2D | Spectral Energy*, *CGWAVE | Spectral Energy*, or by choosing *STWAVE | Model Control*, clicking on the *Boundary Conditions* button to open the Boundary Conditions dialog, and then clicking the *Spectral Energy* button. The spectrum represents energy densities at discrete values over a range of angles and a range of frequencies for a given wave condition.

Tools

- **Select Cell Corner** – Select a cell corner to view or edit the energy value.
- **Pan** – Pan the spectral grid.
- **Zoom** – Zoom in the window.
- **Rotate** – Rotate in the window.
- **Frame** – Frame, or zoom to the extents of the grid.
- **Contour Options** – Bring up the contour options dialog for setting the spectral grid contour display options.

Grid Options

- **Create Grid** – Brings up the Create Spectral Energy Grid dialog.
- **Generate Spectra** – Brings up the Generate Spectra dialog.
- **Import Spectra** – Brings up the Import Spectra dialog.
- **Export Spectra** – Brings up the Export Spectra dialog where you can choose a location to save the spectra.

Spectral Tree Options

- **Spectral Manager Tree** – Select a spectrum in the tree. The selected spectra will be displayed in the Spectral Viewer. The tree's right-click options are described below.
- **Select Spectral Grid** – Brings up a dialog to choose a spectrum. Only one spectrum can be viewed at a time. This selects the spectrum to use/visualize. For models that can only use one spectrum, this spectrum will be tied to the model when leaving the dialog (i.e. clicking "Done").

Grid/Spectra Right-click Options

- **Generate Spectra** – Opens the Generate Spectra dialog. This option is available when the grid in the Spectral Manager Tree is right clicked.
- **Edit Spectra** – Opens the Edit Spectra dialog where the Parameter Settings, Angle Settings and Spectral Parameters can be edited.
- **Export Spectra** – Brings up the Export Spectra dialog where you can choose a location to save the spectra.
- **Rename Grid** or **Rename Data Set** – Rename a grid or spectrum.
- **Delete Grid** or **Delete Data Set** – Deletes a grid or spectrum.
- **Properties** – Brings up the Spectral Grid Properties dialog. This option is available when the grid in the Spectral Manager Tree is right clicked.

View Options

- **Cartesian/Polar View –** View and edit the spectral grid using a Cartesian or polar view.

Graphic Options

- **Selection –** View the Frequency, Angle, and Energy of the selected cell corner. The Energy can be edited for the selected points.
- **Cursor –** View the Frequency, Angle, and Energy as the cursor moves over the grid.

2D Plot Options

- **Frequency Integration Plot –** Turn on to show an energy vs. frequency plot for the selected spectrum.
- **Direction Integration Plot –** Turn on to show a direction vs. energy plot for the selected spectrum.

Related Topics

- Create Spectral Energy Grid
- Generate Spectra
- Import Spectra
- Spectral Grid Properties
- STWAVE Menu
- CGWAVE Menu
- Cartesian Grid Module

Create Spectral Energy Grid

The Create Spectral Energy Grid dialog is accessed through the Spectral Energy dialog (by pushing the *Create Grid* button). A spectral energy grid is created after setting the options. All units for the options are hertz and degrees. The new spectral energy grid will be displayed in the *Spectral Manager* and the *Spectral Viewer* in the Spectral Energy dialog.

Frequency Distribution

- **Number –** Set the number of frequency bands (Number = 30)
- **Delta –** Set the step size (Delta = 0.01) in Hz.
- **Minimum –** Set the minimum frequency (Minimum = 0.04) in Hz.
- **Maximum –** View the maximum frequency (Maximum = 0.33) in Hz.

Angle Distribution

- **Number** – View the number of angle bands (Number = 35).
- **Delta** – Set the step size (Delta = 5) in degrees.
- **Minimum** – View the minimum angle (Minimum = 0.0) in degrees.
- **Maximum** – View the maximum angle (Maximum = 360.0) in degrees.

Return to Spectral Energy

Generate/Edit Spectra

This page describes both the Generate Spectra and Edit Spectra dialogs since they are almost the same dialog. The Generate Spectra dialog is accessed through the Spectral Energy dialog (by pushing the Generate Spectra button). The Edit Spectra dialog is accessed through the Spectral Energy dialog by right-clicking on a grid or spectra in the Spectral Manager and clicking Edit Spectra. SMS creates/edits the spectra when the Generate/Edit button is clicked. All units for the options are feet or meters, depending on the coordinate system. STWAVE runs in metric units, so if the current units are in English units inside SMS, all data is converted to metric when it is saved. The settings are shown below (default values are shown in parenthesis):

Parameter Settings

- **Generation Method** – Choose the method you want to use to generate the spectra. Each method has slightly different options.
- **Replace Old Spectra** – Delete all existing spectra in Spectral Energy dialog after the new spectra are generated. Not available when editing.
- **Directional Spreading Distribution** – Choose to use either the wrapped normal distribution or the cosine power distribution. With the wrapped normal option, a standard deviation and maximum angle cutoff must be specified. With the cosine power option, the spreading index and the maximum cutoff angle must be specified. The recommended cutoff angle is three times the standard deviation of the directional distribution.
- **Gauge Depth** – Water depth in meters (d = 5.0). Choose whether to specify once for all spectra or to specify for each spectrum.

Angle Settings

- **Coordinate System** – The wave direction can either be specified in a shore normal/local coordinate system or a meteorologic/global coordinate system.

Spectral Parameters

The following are used by SpecGen to generate the spectra.

- **Index/Time (hrs)** – Run identifier. If the Generate as Time Steps option is on above it will be labeled as Time (hrs), otherwise it will be labeled as Index. This can be a text string or time stamp. The value is written to the files to identify the spectrum, but is not used otherwise.
- **Angle (deg)** – Approach angle relative to the shore normal in the clockwise direction measured in degrees (wvang = 25.0).
- **Hs (m)** – The incident zero moment wave height (hm0 = 1.0).
- **Tp (sec)** – Wave period in seconds (tp = 20.0).
- **Gamma** – Spectral peak dispersion factor (igamma = 3.3).
- **nn** – Directional or peak dispersion factor, must be even integer (inn = 4).

- **Gauge Depth (m) –** Water depth in meters (d = 5.0). This parameter is available if the Gauge Depth option above is set to Specify for each spectrum.

Spreadsheet Options

- **Import/Export –** Import/export an ASCII, space delimited text file with the spectral generator parameters. The file format is:

SPECTRAL TABLE	Values
Method Option Time Index Angle Hs(1) Tp(1) Gamma (1) Hs (1) Tp(1) Gamma(1) Hs(2) Tp(2) Gamma(2) Wind Fetch nn StdDev Depth	Headers
0 -1 999.0 None 25.0 1.0 20.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	1st row of values
0 -1 999.0 None 30.0 1.0 16.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	2nd row of values

Additional format description is found in CMS-Wave Spectral Table File.

- **Import from Genesis –** Imports wave parameters from the Filtered Ocean Conditions dialog. This button is only available if data has been filtered in Genesis.
- **Spectral Defaults –** This opens a section of the dialog that lists the approximate spectral parameters. Double click a row of values to replace the selected spreadsheet row(s). The period (T), gamma, and nn will be replaced for the row. If the period in the spreadsheet does not match a period in the table, the spreadsheet period is rounded to the nearest table period.

Return to Spectral Energy

Import Spectra

The Import Spectra dialog is accessed through the Spectral Energy dialog by clicking on the *Import Spectra* button. This dialog is used to import existing spectra into the project.

Options

- **Select Spectral Energy File –** Click on the folder icon to browse to the spectral energy file you want to import.
- **Create New Spectral Grid –** If this option is selected, SMS will import the spectra with the grid definition contained in the spectral energy file.
- **Select Existing Spectral Grid –** If this option is selected the datasets are imported as datasets of the selected grid (the grid selected in the combo box below).
- **Import as Time Steps –** Imports the datasets using the identifier as a time value.

File types

The following types of files can be imported through this dialog.

- ***.eng** – Spectral energy file in the CMS-Wave format.
- ***.obs** – Observation output spectra for CMS-Wave.
- ***.dws** –- The BOUSS-2D spectra file format.

Note: The *.h5 spectra files can be opened through the main SMS open command or drag-and-drop to the graphics window. After opening a *.h5 spectra file, you will need to click the Select Spectra to Use button in the Spectral Energy Dialog to select the output spectra.

Related Topics

- Spectral Energy Dialog

Spectral Grid Properties

The Spectral Grid Properties dialog is accessed through the Spectral Energy dialog by right-clicking on a grid in the Spectral Manager tree and clicking on Properties. This dialog only displays information. The values cannot be changed. If different values are desired, a new grid must be created through the Create Spectral Energy Grid dialog.

Frequency Distribution

- **Number** – View the number of frequency bands.
- **Delta** – View the step size in Hz.
- **Minimum** – View the minimum frequency in Hz.
- **Maximum** – View the maximum in Hz.

Angle Distribution

- **Number** – View the number of angle bands.
- **Delta** – View the step size in degrees.
- **Minimum** – View the minimum angle in degrees.
- **Maximum** – View the maximum angle in degrees.

Return to Spectral Energy

2.12. Datasets(VTK)

Datasets VTK

Datasets

A dataset is a set of values associated with each node or cell in a geometric object. Datasets:

- Can be scalar (1 value) per object or vector (2+ values per object).
- Can be steady state (constant through time) or transient (values change at specified times).
- Can have active information to specify that specific nodes or cells are inactive in a model (generally used by solutions to indicate dry areas). Activity can be represented by NULL values in the dataset or as separate on/off values for nodes or cells.
- Control how contours, vectors, and functional surfaces are displayed.

Generating Datasets

Datasets can be generated in a variety of ways such as:

- Output from a numeric engine (water level, velocity, concentration, transport, etc.)
- Tabular values in a text file entered by the user or exported from another application such as a GIS
- Created by interpolating from a scatter point set to a grid, or mesh
- Generated by performing mathematical operations on existing datasets with the Dataset Calculator

Project Explorer

Datasets are displayed and managed in the Project Explorer. See the Project Explorer article for more information.

Active Dataset

Each geometric object has a dataset that is termed the "active dataset." The active dataset is used for contour and vector display and may be used for functional surfaces, 2D plots, or other functionalities keying off this dataset.

Project Explorer Icons

Different icons are used to represent datasets in the project explorer including the active/inactive state of the dataset. These icons are below:

Dataset Type	Inactive Icon	Active Icon
Elevation		
Scalar		
Vector		

Time Information

Transient datasets (those that change with time) have information and functionality that is not available for steady state datasets. A time step represents a specific time and its values in a transient dataset. A dataset may use absolute times meaning the dataset has full date/time information. Alternatively, a dataset can use relatives times which means they know how much time has elapsed since some non-specified zero time (generally the beginning of the simulation). When transient datasets are present, the Time step window may be present depending upon the current time settings as specified in the preferences dialog.

Folder

The datasets and solutions are organized by folders. The user can create new folders and move datasets, solutions, and folders to other folders anywhere on the *Project Explorer*. Folders can be created by right clicking on the certain items in the *Project Explorer* and selecting *New Folder* in the menu. A dataset or folder can be deleted simply by selecting the folder and selecting the Delete key or by right-clicking on the item and selecting the Delete option in the corresponding pop-up menu.

Datasets on VTK Objects

Datasets as used on VTK objects have different functionalities then those used in the original geometric representations in SMS. Some of the differences for VTK datasets include:

- Each dataset can be mapped either to nodes or cells and the same geometric object can have both types of datasets at the same.
- A new dataset calculator has been created. The new calculator has additional functionalities and options. For more information see the Dataset Calculator VTK topic.

Related Topics

- Layout of the Graphical Interface

Conversions Scalar↔Vector

Conversions Scalar↔Vector

Scalar to Vector

Converts two scalar datasets to a single vector dataset. The specified scalar datasets can be either magnitude and direction or x and y components. To convert to vector do the following:

- In the project explorer, select two scalar datasets.
  - Click on the first dataset, press and hold down the *ctrl* key, then click the second dataset.
- Right click on the selected datasets and select *Scalars to Vector*.
- Select whether the datasets are magnitude and direction or x and y components.
- Give the new vector dataset a name in the *Dataset Name* field.
- Click *Ok*.

Vector to Scalar

Converts a single vector dataset into one or more scalar datasets. The resulting scalar datasets can be magnitude, direction, x or y components. To convert to scalar do the following:

- In the project explorer, right click on a vector dataset and select *Vector to Scalars*.
- Right click on the selected datasets and select *Scalars to Vector*.
- Place checkbox's next to the datasets that you want to create.
- Give the new scalar datasets a prefix in the *Prefix* field.
- Click *Ok*.

Dataset Calculator VTK

The dataset calculator allows you to create new datasets using mathematical expressions. The inputs for the mathematical expressions can be user defined constants, existing datasets, data derived from the geometry, or data derived from datasets such as gradients or activity information (0/1).

The dataset calculator expressions can include:

standard operations: + - * / ^ .

build unit vectors: iHat, jHat, kHat (ie (1,0,0), (0,1,0), (0,0,1))

abs

acos

asin

atan

ceil

cos

cosh

exp

floor

log

mag

min

max

norm

sign

sin

sinh

sqrt

tan

tanh

Expressions

In the top left of the dialog, there are fields for name and expression. The name specified will become the dataset name. The expression defines the mathematical operation that will take place. You can insert a predefined function into the expression field using the "Insert f(x)" combo-box. The specified function will be pasted into the current cursor location in the expression field and will include an indication of the number of values expected.

SMS maintains a list of expressions currently defined. Specifying a name and expression and clicking "Add" will add an expression to the list of expressions. If you select an expression from the list of expressions, the name and expression fields will be populated. You can make the current list of expressions the default list by saving settings ("File | Save settings" from the main menu). You can move your expressions to another computer or user by using the save and load buttons.

Variables

When the expression field is modified, SMS will parse the expression to find variable names in the expression. A variable must start with a character but can include digits after the first character. "Var1" would be valid but "1Var" would not. The list of variables determined by SMS will appear in a spreadsheet in the top right of the dialog. Each variable has a name, type, and source information. The name is parsed from the expression field and cannot be edited in the spreadsheet. The type can be chosen from the spreadsheet. The source information defines the options used which vary by variable type and cannot be edited in the spreadsheet. Beneath the variables spreadsheet are controls to specify the source information for the currently selected variable. The controls that will be available will depend upon the variable type.

The simplest variable type is a value. While it is possible to include numeric values in the expression field, it can be more clear to name specific variables. For example, you may want to have an expression use a variable named "gravity" that is specified as a value variable. This makes the expression more readable and helps remind people to change this if working in a different set of units or similar situation.

Another type of variable is a dataset. This will use the specified dataset anywhere the variable shows up in the expression. Note that both scalar and vector datasets can be used in an expression.

The final type of variable are "Derived" variables. Derived variables can be based upon one of three different sources. They can be based upon node geometry, cell geometry, or datasets. Derived nodal options include location (3D vector) and nodal spacing (average spacing to neighboring nodes). Derived cell options include area, centroid (3D vector), extents minimum (3D vector), extents maximum (3D vector), and perimeter. Derived dataset values include activity (0/1 for each node/cell for each timestep), directional derivative (vector representation of the gradient), and vector angle (in the cartesian coordinate system used in math).

Output Location

Datasets in VTK can be associated with nodes or cells. When using the dataset calculator, you can specify whether you want the dataset created to be associated with nodes or cells. If you don't specify the output location, SMS will decide based upon the variables used in your expression. With the exception of values, each variable will have an affinity to nodes or cells. If any variable has affinity to nodes, SMS will default the output to be based upon nodes. If all the input variables have an affinity to cells, the output will be a cell based dataset. The variable affinity is generally intuitive: dataset variables use dataset, derived variables come from whether they are node, cell, or dataset derived. One exception to the intuitive rule is "directional derivative" which is derived from datasets. This variable will have an affinity opposite of the input dataset. If the input dataset is nodes, the directional derivative will be computed on cells.

Whenever input data has an affinity opposite to the output location, the data is converted before the expression is evaluated. Generally, this means that the value for each node or cell (whichever conversion is taking place) is determined by averaging the surrounding values (connected cells or nodes belong to the cell). The exception to this rule is activity which is never averaged. A node is considered active if any of the surrounding cells is active.

Working with Scalars/Vectors

As mentioned previously, an expression may contain both scalar and vector components. The result of an expression may also be either a scalar or vector dataset. Some sub-expressions can be used with either type of dataset. For example, you can multiply a value by either a scalar or vector dataset. Each component in the vector dataset will have the multiplication applied. Other sub-expressions only make sense when dealing with a certain type of input dataset. The "." operator can be used to compute the dot product of two vectors and doesn't make sense for scalar datasets. The identifier iHat can be used to create a unit vector in the x direction (1.0, 0.0, 0.0). You can use the dot operator to extract the x component of a vector using an expression like: "myvector.iHat". jHat and kHat can be used similarly to extract y and z components. You can also convert components into vectors using statements similar to: "vx*iHat + vy*jHat." The mag function extracts the magnitude of a vector and also does not apply to scalar datasets.

Output Times

By default the output dataset will have times corresponding to any time used in any of the input datasets. If all the input datasets are steady state datasets, the output dataset will be steady state. You can specify specific output times for the output dataset. This can be useful to reduce the number of timesteps that would be generated or focus on a range of times. If all the input datasets are steady state and output times are specified, the resulting dataset will have multiple timesteps each with the exact same values. If an input dataset doesn't have a timestep at an output time, the data will be interpolated between the nearest timesteps. If the time occurs outside the range covered by the dataset, the nearest dataset time will be used.

Errors

If there is a problem with the expression entered, SMS will give an error message and allow you to correct the problem. If the problem is with the expression itself, the message should indicate about where the problem exists. The position may be off slightly so examine the whole expression carefully if the cause isn't immediately apparent.

Interpolation VTK

VTK Dataset Interpolation

VTK datasets can be interpolated to create functions or datasets on mesh2d, cgrid, scatter, VTK mesh or curvilinear geometric objects. The interpolation is invoked in the project explore by right-clicking on the VTK mesh or Curvilinear geometric object (from which the source datasets will be obtained) and then selecting the *Interpolate to...* option.

Interpolation Dialog

When the user selects an interpolation command, the interpolation option dialog. The user selects the appropriate options and once the OK button is selected, the interpolation procedure is performed. The user specified options include:

- Interpolation Method – The user selects a current method that is used for all interpolation until the user selects another method. The supported methods include:
  1. Standard (Linear)
  2. Inverse Distance Weighted (IDW)
- Extrapolation Method – If the VTK dataset does not bound the data being interpolated to, an extrapolation value is used for each location outside the boundary or if the location has be marked as inactive. The user selects the method to be used to generate the extrapolation value from the following:
  1. Inactive – The dataset value is set to a null indicator and is not displayed. (note - scatter sets do not support inactive cell so this option is not available when they are the selected target.)
  2. Inverse Distance Weighted (IDW)
  3. Constant Value – A single value is applied to all extrapolated locations in the dataset.
  4. Existing Dataset Value – The corresponding value from a specified existing dataset can be used for locations outside of the bounds of the source VTK dataset. The dataset must be from the same object being interpolated to and must be of the same type (i.e. scalar, vector).
- Target objects – The use selects a target for interpolation from this tree list of geometrics objects.

- Source object info – This section contains a tree list of the source datasets and information regarding their usage as follows
  1. Datasets to interpolate – The user selects the datasets to interpolate from by marking their check box. Multiple datasets may be interpolated at a time.
  2. New name – The user may rename the dataset created as a result of the interpolation.
  3. Map Z *ndash; The user may designate the dataset created as the elevation or "Z" dataset by marking the appropriate check box.
  4. Extrapolation Constant Value – The column is displayed when the "Constant Value" Extrapolation Method is selected. The user must enter the single value to apply to the extrapolation locations for each dataset to be interpolated from.
  5. Extrapolation Dataset – This column is displayed when the "Existing Dataset Value" Extrapolation Method is selected. The user must select an extrapolation dataset for each dataset to be interpolated from.

Related Topics

- Scatter Interpolation

3. Modules

Modules

SMS Modules

The commands in SMS are divided based on the types of data they operate on. As you switch from one module to another module, the Dynamic Toolbar and available menu commands change. This allows you to focus only on the tools and commands related to the data you are currently working with in SMS. Switching from one module to another can be done instantaneously to facilitate the simultaneous use of several data types when necessary. Only one module is active at any given time. However, the data associated with a module (e.g. a 2D finite element mesh) is preserved when the user switches to a different module. Activating a module only changes the Dynamic Toolbar and available menu commands.

Module Selection

There are several ways to switch from one module to another. These include:

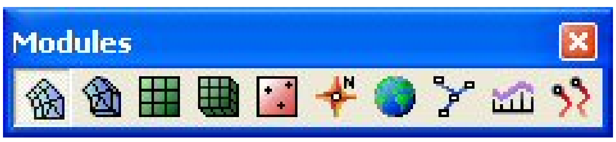
- Select an entity in the *project explorer*. The module containing the active entity becomes active.
- Right click on the *project explorer* and select the *Switch Module* command.
- Click on the module icon in the module toolbar. The module toolbar is displayed at the bottom of the project explorer by default.

(Note: Switching modules should not be confused with changing the current model inside of a module. When a new model is selected, the tools and menus may change, and the data will be converted as much as is possible. However, some data may be lost.)

Modules in SMS:

- Mesh Module
- Cartesian Grid Module
- 3D Cartesian Grid Module
- Scatter Module
- Map Module
- GIS Module
- 1D River Module
- 1D Grid Module
- Particle Module
- Images
- CAD Data

Module Toolbar



Module Toolbar

The Module Toolbar is used to switch between modules. Only one module is active at any given time. However, the data associated with a module (ex. a 2D finite element mesh) is preserved when the user switches to a different module. Activating a module simply changes the set of available tools and menu commands.

Annotations

Annotation Objects

The XMS application family provides a series of tools to annotate the data in an application for presentations, animations and screen shots.

These tools (annotation objects)are accessed through the Annotations Module and include:

- Images
- North Arrows
- Scale Bars
- Text
- Lines
- Ovals
- Rectangles

Screen vs World Space Layers

All annotation layers either contain objects referenced to world or screen coordinates. Objects referenced to world coordinates will change size and position on the screen with the underlying data. This is useful to identify specific locations in your model such as peir locations. Objects associated with screen coordinates do not move on the screen with the underlying data. This is useful for titles, legends such as north arrows and scale bars, and logos. Some types of annotations can only be created in screen space layers including North Arrows, Images, and scale bars.

If the first annotation object you create, could be part of a screen or world space layer SMS will ask which type of layer you wish to create and add the object to. You can create additional layers by right clicking on the *Annotation Data tree* item and selecting ***Create Screen Space Layer*** or ***Create World Space Layer***. Layers are differentiated by including an 'S' for screen space layers or 'W' for world space layers in their icons in the project explorer.

If multiple layers exist, any newly created annotation object will be placed in the "current" layer.

Annotation Object Attributes

The extents of annotation objects defined by a frame. The user defines this frame initially when creating the annotation object by left clicking at any point on the screen and dragging a rectangle with the mouse (left button still down). The display will show the frame while you drag with the mouse. (Points and lines defining degenerate frames are not allowed.) When the user creates a annotation, if the frame is too big for the window, it will be resized appropriately. Annotations can't be resized or moved even partially outside of the borders of the window. If the user resizes a annotation through a quick mouse drag and the cursor lands outside the window, the annotation will be redrawn to take up all the window space in that direction.

This frame bounds the region of the screen where the object will appear with the modeling data. The user interacts with the object by interacting with its frame and specifying its attributes or properties (see the section on selection below). The frame anchors the annotation object on the screen. This anchoring defines both the size and position of the object. The x-location, y-location, x-size and y-size are all defined independently as either a pixel value or percentage of the screen.

Horizontally, the user can position the left edge, the right edge or the center of the object. If the user positions the left edge, the object position is defined relative to the left edge of the screen. If the user positions the right edge, the object position is defined relative to the right edge of the screen. If the user positions the center of the object, the object position is defined relative to the horizontal center of the screen.

For example, the left side of the frame may be specified as 100 pixels from the left edge of the screen. Alternatively, the user may specify that the right edge of the frame should be 10% of screen width from the right edge. Finally, the user may specify that the center of the object is 100 pixels to the right of the center of the screen.

The vertical position and sizes of the object are similarly specified in the anchoring attribute of the object.

All annotation objects also have attributes. The specific attributes depend on the type of object. The attributes define color, line thickness, fill properties, associated images, etc.

Screen Space Images

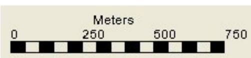
A screen space image is simply a graphics icon mapped to the screen. A typical application would be to display a company, department, or municipality logo next to the numeric model being displayed in the graphics window.

Attributes of the screen space images include:

- General anchoring attributes
- The image file being displayed as part of the project
- Whether the image is being displayed as a scaled (distorted object), scaled based on its original aspect ratio, or locked at another aspect ratio.
- Transparency – The image properties dialog have a transparency checkbox. When checked it will cause the image to be redrawn with the most used color in the image. When that it is checked, it also causes the color checkbox and the tolerance edit field to become available. If the color checkbox is checked, it will activate the color button and the color button will have the latest chosen image color painted on it or the most used color in the image, if it has not been activated before. Clicking on the down arrow part of the color button causes a color popup to be displayed with swaths of the 40 most used colors in the image or all the colors in the image, if the image has less than 40 colors. Clicking on one of those colors will cause the image to be redrawn with that color made transparent in the image. The tolerance edit field allows for variation in the matching of the red, green and blue components. The tolerance field ranges in allowable values from 0.0 to 1.0. 0.0 means the red, green and blue components must exactly match. Values higher than 0.0 indicate the degree of variation from the given color. Clicking the transparency checkbox to the off state causes the image to be redrawn with no transparency.

Scale Bars

A scale bar occupies a fixed size of the screen to display the relative size of the objects in the simulation. The user defines the minimum width of the scale bar section (in pixels), along with a minimum and maximum height of the scale (also in pixels). The XMS application adds a "Units" label (meters in the image shown below) and labels for the model distance related to the scale divisions.



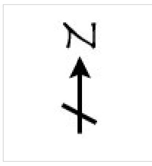
The XMS applications will compute a well conditioned number to use as the scale increment that fits in the specified scale bar extents.

Attributes of the scale bar include:

- General anchoring attributes.
- The minimum spacing between distance labels.
- The minimum division width (in pixels). The XMS application determines the number of divisions based on the minimum divisiion width and the width of the frame.

- The minimum and maximum height of the scale bar object.
- The font (style, color, size) used to label the scale bar.
- Whether or not the area behind the scale bar will be filled, and if so, with what color.

North Arrows



North arrow objects consist of automatically rotating screen space images. When an XMS application is installed, at least one default North Arrow image will be included in the application's home directory. Users may create or download as many north arrow icons as desired. These icons are displayed at the specified location (anchored with the standard options), but will rotate as the view direction changes so that the "up" direction of the icon always aligns with the "North" or positive "Y" direction.

Attributes of north arrows include:

- See screen space image attributes.

Text

Text can be created in world or screen space layers.

You can enter text by clicking in the graphics window with the create text tool active.

You can set the following attributes for text:

- Font – including size
- Color
- Background – fill behind with the background color or another color
- Border and border thickness

Lines/Arrows

You can create Lines/Arrows using the Create Line Tool. Lines/Arrows can be created in screen or world space layers.

The attributes available for Lines/Arrows include:

- Type – Dashed or solid w/ thickness
- Color
- Arrowheads – location (beginning, end, both) and size

Ovals

You can create ovals by dragging a box with Create Oval tool active. Ovals can be created in world or screen space layers.

The attributes that can be specified for ovals include:

- Line type – dashed/solid with width
- Line color
- Fill and color or No Fill

Rectangles

Rectangles can be created in screen or world space layers. Their attributes are the same as those available for ovals.

Selection

The first toolbar button is used to select and set attributes for annotation objects. This requires that objects exist to be selected. In this case when you press this tool and then left click in the annotation object, the object frame will be drawn around the annotation. In addition to the frame, the XMS application displays grab handles on the corners and edges of the frame. The user modifies the rectangular shape of the annotation by dragging one of the grab handles and changes the position of the object by dragging the annotation (click at any point in the object interior).

When this tool is active the right mouse button will pop up a menu of all dialogs for that particular type of annotation. This will include at least "Attributes" which will display a dialog of the objects attributes for editing.

Viewing Annotations At Specific Time Intervals

Available in SMS 11.1 and in development for GMS, annotations can be setup to be viewed at specific time intervals. To setup annotations so they only are displayed at specified time intervals do the following:

- Right click on the Annotation layer in the tree then select *Properties...*.
- This dialog will display the *Annotation Layer Properties* dialog.
- Check the *Apply time range* checkbox
- Modify the "begin" and "end" time controls to specify the range for when annotations are visible.
- Click *Ok*.

Now film loops and time steps will only display the annotation when its within the range specified.

3.1. 1D Grid Module

1D Grid Module

The 1D grid module contains an interface for the GenCade shoreline morphology model.

1D Grid Module Tools

These tools allow constructions of a 1D grid, shorelines associated with the grid, and structures such as seawalls, groins and breakwaters associated with that shoreline.

- **Select Point**
- **Create Point** – Create a point along the coastline.
- **Select Detached Breakwater**
- **Create Breakwater**
- **Select Jetty or Groin**
- **Create Jetty or Groin**
- **Select Seawall**
- **Create Seawall**

See 1D Grid Module Tools for more information.

1D Grid Module Menus

The following menus are available in the the 1D Grid module.

- Standard Menus – See Menu Bar for more information.
- Data – See Scatter Data Menu for more information.
- GenCade

Related Topics

- 1D Grid Display Options
- Modules
- GenCade

1D Grid Display Options

The properties of all 1D Grid data that SMS displays on the screen can be controlled through the One Dimensional Grid tab of the Display Options Dialog. This dialog is opened by right clicking on the GenCade Data entry in the Project Explorer and selecting the Display Options command. (It can also be accessed from the from the Display menu or the display options Macro.)

The entities associated with the 1D Grid module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available one dimensional grid display options include the following:

Grid Objects

- Y Scale Factor – magnification in the direction perpendicular to shore
- Grid display
- Initial Coastline display
- Current Coastline display (based on dataset)
  - Coastline points
- Minimum/Maximum Coastline and Envelope (zone covered by transient coastline and extremes based on dataset)
- Reference Coastline

Structure Objects

- Sea Walls
- Groins
- Breakwaters
- Bypass Cells
- Beach Fills

Return to 1D Grid Module

1D Grid Tools

The 1D grid tools are available in the 1D grid module when working with the GenCade model. These tools allow the user to edit the features on a 1D grid. These festures include:

- Shorelines associated with the grid
- Structures along the shoreline:
  - Seawalls
  - Groins
  - Breakwaters

It is recommended that you manage a GenCade project through a conceptual model in a GenCade coverage. The conceptual model allows more flexibility when specifying the structures because it works in real world space. These tools work on the 1D grid using grid cell indices and distances from the grid to locate the objects. The GenCade menu also provides commands to edit the objects.

The tools include:

- **Select Point –** This tool allows the user to select a point on the initial shoreline defined for the grid and drag it closer to or farther away from the grid. Since the distance along the grid is not variable for the selected point, the edit only affects the local "Y" value of the grid point. Since the grid usually consists of many points along the shoreline, editing the shoreline with this tool can be tedious.
- **Create Point –** This tool is currently disabled. With other 1D grid models that have been supported in the SMS package, creating grid points using this tool was supported. The only 1D grid model currently included in the package (GenCade), does not support this feature. This tool would be used to create a point along the initial coastline.
- **Select Detached Breakwater –** This tool is used to edit a breakwater positioned along the grid. With this tool active, a click in the graphics window selects an endpoint of a breakwater, and dragging the mouse with the end point selected moves the endpoint of the breakwater. The depth, transmission and permeability of the breakwater must be assigned using either the GenCade|Detached Breakwaters command or by assigning these attributes to an arc in the conceptual model.
- **Create Breakwater –** This tool can be used to create a breakwater in a simulation. Clicking in the graphics window with this tool active defines a starting point for a breakwater. SMS will then draw a "rubber band line" from this location to the cursor until a second location is clicked, terminating the breakwater. Attributes for the breakwater must be assigned using either the GenCade|Detached Breakwaters command or by assigning these attributes to an arc in the conceptual model.
- **Select Jetty or Groin –** This tool is used to edit the length of a groin or jetty positioned along the grid. With this tool active, a click in the graphics window selects an endpoint of a groin , and dragging the mouse with the end point selected moves the endpoint of the groin, thus changing its length. Other attributes of the groin/jetty must be assigned using either the GenCade|Groins or GenCade|Inlets commands or by assigning these attributes to an arc in the conceptual model.
- **Create Groin –** This tool can be used to create a groin in a simulation. Clicking in the graphics window with this tool active defines the end point for a groin. SMS will connect this location to the grid defining the groin. Attributes for the groin must be assigned using the GenCade|Groins command or by assigning these attributes to an arc in the conceptual model.
- **Select Seawall –** This tool is used to edit the shape of an existing seawall along the grid. With this tool active, a click in the graphics window selects a point in a seawall. Dragging the mouse with the point selected modifies the seawall. As with grids, seawalls often include multiple segments making them tedious to edit using this

approach.

-  **Create Seawall –** This tool can be used to create sea wall segments along the grid. With this tool active, a click in the graphics window starts the creation of a segment. A second click terminates the segment. When the segment is complete, SMS determines if this segment overlaps existing segments and trims the existing segments to the new segment if an overlap exists.

Related Topics

- 1D Grid Module

3.2. 1D River Module

1D River Module

The purpose of the River Module is to provide tools to perform one dimensional (Step Backwater) hydraulic modeling within SMS. The module include tools to process digital terrain (scattered datasets) and map data (coverages) to build the basic geometry necessary for a 1D Hydraulic Model. Much of the information for developing models with these tools is described in the information on River Coverages in the Map module.

The 1D River Hydraulic tools are used to create, edit and apply networks of river reaches and cross-sections. The created networks must be associated with a specific 1D model (currently the only major model supported is HEC-RAS. The module includes tools for visualizing results computed for cross sections and reaches.

The general process for developing a model consists of the following steps:

1. Preparing a background digital terrain model that represents the river channel bathymetry and surrounding flood plain with enough detail to substantiate the modeling objectives.
2. Developing a 1D-Hydraulic Centerline coverage including the centerline and optional bank arcs.
3. Creating the cross section arcs at important/required locations along the section of river being modeled.
4. An Area Property coverage can be used to map roughness values to line properties on the cross sections
5. Extracting cross sections from the TIN and establish the 1D Model
6. Export the GIS data and finish defining HEC-RAS (or other models).

Alternatively, It is possible to establish the hydraulic model without extracting cross section information from a TIN. Cross sections which have already been surveyed can be assigning to an arc. This, along with geo-referencing the data is done using the cross section editor from the River Tools menu in the Map module.

The heart of the 1D River Hydraulic Module is a database of cross sections. A set of tools called the Cross-Section Database Manager allow the user to edit and manage the cross sections. All cross sections referenced from the 1D River Hydraulic Module or the Map Module must be part of this database. Cross sections may be added to the database by importing data from an external source, hand editing in the Cross-Section Database Manager, or extraction from topographic data in the Map Module.

River networks are displayed as a schematic of reaches and cross sections. Networks may be georeferenced or purely symbolic. Each cross-section is represented by a cross-section icon displayed at the location of the cross section if the section is georeferenced, and reach icons, which are displayed on top of the first cross section in each reach. The icons can be moved according to the user’s preference by selecting the icon and dragging it to the desired location.

The cross-section attributes can be changed by using the Select Cross-Section Tool. Double- clicking on the desired cross-section opens the River Cross Section Atts Dialog which can be edited similar to the Map Module. The only difference is that the reach the cross-section is attached to cannot be changed. Likewise the reach attributes can also be changed within the 1D River Hydraulic module.

1D River Module Tools

See 1D River Module Tools for more information.

Scatter Module Menus

See 1D River Module Menus for more information.

Related Topics

- 1D River Module Tools
- Area Property Coverage
- 1D River Conceptual Model
- HEC-RAS
- 1D River Hydraulics Data Browser
- RivHyd Data Export
- RivHyd Post Processing
- Modules

1D River Hydraulics Data Browser

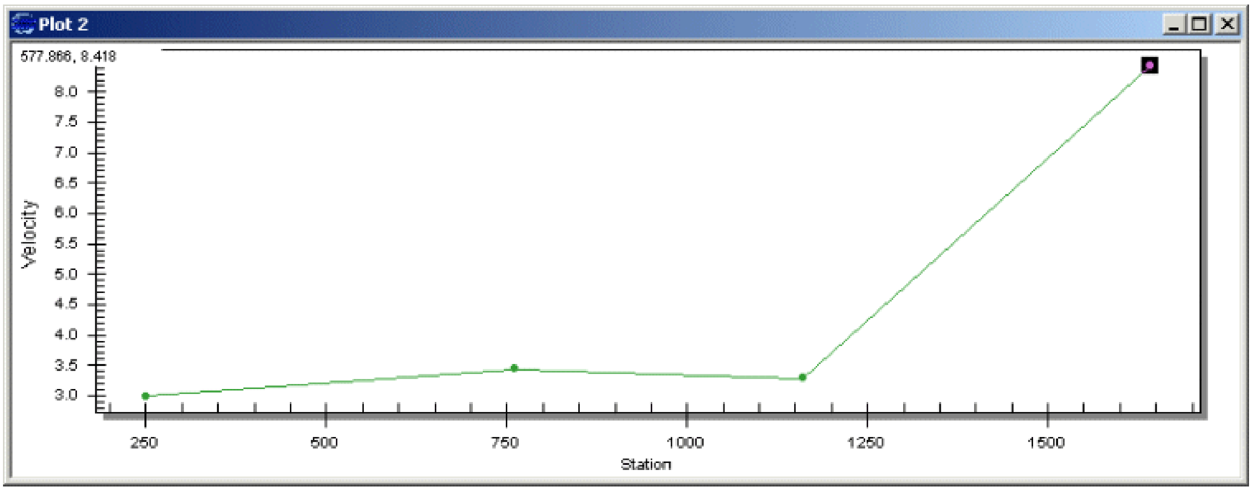
The top list box shows the 1D River Hydraulic solutions. A solution is a list of functions. When a solution file from a numerical simulation is read in, the functions may be added to an existing solution or to a new solution. New solutions can be created and existing solutions deleted with the buttons on the right of this box. Solution names (except for the Generic solution) can be changed by clicking on desired solution and typing the name you want it to have. The box in the middle of the browser displays the scalar datasets in the currently selected solution. The names for these functions may be changed in the same manner as the solutions names. Datasets may also be moved to other solutions. The bottom box shows the time steps in the currently selected dataset. Clicking on the info button will bring up a dialog box which displays information about the selected time step of the selected dataset.

Back to 1D River Module

1D River Hydraulics Profile Plots

A 1D profile plot can contain either the currently selected river reaches or specific reaches specified in the second frame of the plot wizard. In addition, the plot may set to show the currently active dataset and time step, or the user may specify specific datasets and time steps.

The plot shows the dataset values at each cross-section along the reach. If values vary across the cross sections, the plot may be set to show the minimum, average, and/or maximum values at the cross section.



Return to Plot Window

1D River Module Menus

The following menus are available in the the 1D River Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data
  - Create Visualization Tin
  - Rivhyd to Scatter

Model Specific Menus

- HEC-RAS

1D River Module Tools

The River Tools is not necessarily a model, but provides the tools and commands necessary to build 1D Hydraulic modeling data from feature object coverages. The River Tools primarily support the HEC-RAS model, but will be used to support additional hydraulic models in future versions.

The River tools provide the ability to define a hydraulic model using a 1D-Hydraulic Centerline coverage and a 1D-Hydraulic Cross-section coverage. The layout of the feature objects defining the centerline and cross sections establishes the direction, the stationing, and the topology (connectivity between cross-sections) of a hydraulic model. Further, using these two coverages cross sections may be automatically extracted from a digital terrain model and then edited, merged, or combined with other cross section information to provide the geometric basis of the model. An area property coverage can also be used to map materials (Manning's roughness coefficients) to the cross section based on some type of aerial distinction (land use or soils) that may be available. The diagram below illustrates how these coverages are used to establish a hydraulic model.

The river tools also allow you to interpolate cross sections to establish more cross section information in between surveyed or extracted cross sections.

Results data from HEC-RAS and other hydraulic models can be read back in and used to perform a flood plain delineation. The flood plain delineation algorithm in SMS works better with a denser set of resulting water surface elevation points and so there are river tools that allow a water surface elevation computed at a cross section to be interpolated (copied since it will be the same value) along cross section arcs, or along a centerline.

- **Select Cross-section**
- **Select Centerline**

Related Topics

- 1D-Hydraulic Cross-section
- Area Property Coverage
- Extracting Cross Sections
- Editing Cross Sections
- Interpolate Cross Sections
- Managing Cross Sections
- Recompute All Stations
- 1D River Module
- Dynamic Tools

Delete Simulation

Deletes the link SMS maintains to the current HEC-RAS simulation project file.

Related Topics

HEC-RAS

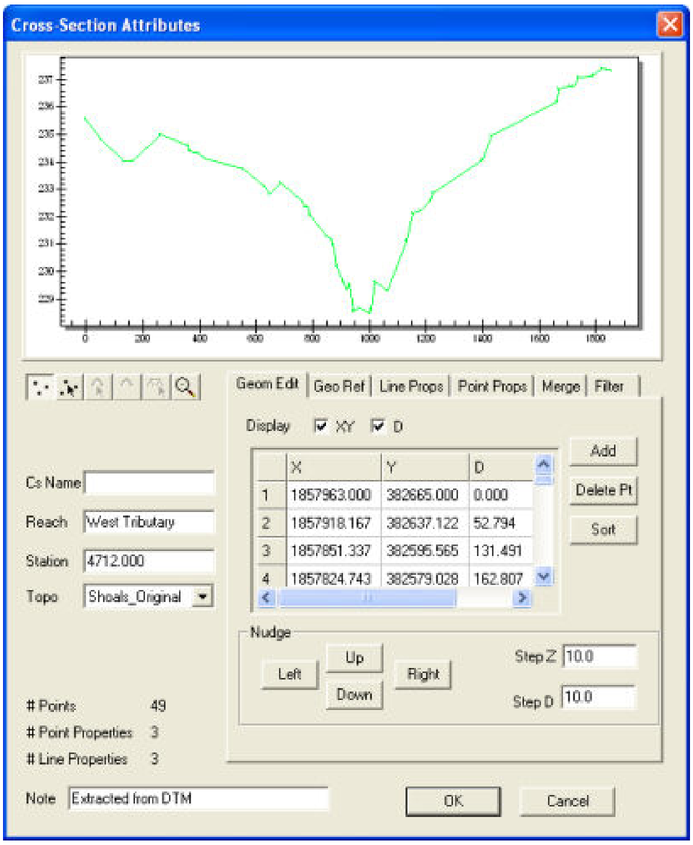
Editing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. However, extraction of cross sections from digital terrain models is not the only way that they can be created, nor is extraction always the only thing that needs to be done. For example other ways cross sections can be entered for use include: including importing from a spreadsheet, or entering manually. In such cases, and many times after extraction from a digital terrain model there are edits that must be performed in order to prepare the cross sections for hydraulic modeling.

You can edit cross sections in one of three ways:

1. If you double click on an arc in a 1D Hydraulic Cross Section coverage you can assign a cross section from a database. After assigning the cross section you can also enter the editor for that cross section.
2. You can open a cross section database for editing (or create a new database) using the Manage Cross Sections command.
3. You can also open an existing cross section database using the *File | Open* command.

The operations described in the following paragraphs can be done using the cross section editor shown in the figure below.



General Properties

In order to identify information about the cross section in the database a name (not required), a reach, a station, and the name of the topographic data used to extract the cross section (if applicable) can be defined. A note about the cross section can also be defined. Not all of these attributes are critical for the development of a hydraulic model, but they are useful in managing the cross section within a database.

Editing Geometry

Cross section points can be added, or values edited when the Geom Edit tab of the editor is active. XY values are available when the actual 3D position of each point on the cross section is known. The more traditional D-Z pairs define the distance from the starting point and a corresponding elevation.

Geo-Referencing

Geo-referencing information provides the spatial (x-y) location of the cross section and included geometry. This information is inherent in the 3D coordinates, when extracting cross sections from a digital terrain model. However, if the cross section geometry is taken from a survey then the actual x-y-z coordinates of the points may not be known. In order to use the data within SMS for flood plain delineation, a proper geo-reference must be provided.

A cross section can have one of the following georeferencing definitions: All points specified (i.e. extracted cross sections will be of this type), Use two points (i.e. the coordinates of the beginning and ending location along the cross section defined), Use one point an angle (i.e. the centerline location is known and some angle relative to it defined), or no geo-referencing defined.

The geo-referencing is defined from the Geo Ref tab in the cross section editor.

Line Properties

Line properties define segments of material properties along the cross section. When using an area property coverage during extraction from a digital terrain model these properties are automatically marked and defined. However, they can also be established manually from within the Line Props tab in the cross section editor.

Point Properties

Point properties include thalweg, left bank, and right bank (other properties can be defined but are not mapped/saved to hydraulic models from within WMS) locations. When using a centerline and bank line arcs from a 1D Hydraulic Centerline coverage during extraction these points are marked. You can also have WMS "Auto Mark" these points by looking for the lowest elevation (thalweg), and appropriate breaks in elevation/slope (banks). Point properties are edited from within the Point Props tab in the cross section editor.

Merging

It is possible to combine a surveyed cross section with a section extracted from a terrain model for the flood plain (i.e. the terrain model does not contain enough detail to define the cross section of the river) using the tools in the Merge tab in the cross section editor. Two different cross sections can be merged, with rules for locations and precedence defined in order to create a new cross section.

Filtering

It may be that there are more points defining the cross section than are necessary (or that the hydraulic model is capable of processing). The Filter tab in the cross section editor allows you to specify rules for filtering "insignificant" points along the cross section. This can be particularly important when extracting cross sections from a dense digital terrain model.

Related Topics

- 1D River Module Tools

Extracting Cross Sections

The *Extract Cross-sections* command uses the cross section arcs and a digital terrain model (TINs are the only source that can currently be used) to extract the elevations at vertices of the feature arc cross-sections, or at the intersection points with the triangles.

Cross-sections for individual arcs may be extracted by selecting the arc(s) before choosing the *Extract Cross-sections* command. If not cross-sections are selected then the *Use All Cross-sections* option is used.

Point properties (thalweg, left bank, right bank) can be defined from a 1D-Hydraulic Centerline coverage, or by AutoMark. The AutoMark option will examine the elevations of the extracted cross sections and try to infer the thalweg (low point) and the left and right bank points (change of slope) automatically.

Line properties can be determined from an area property coverage by intersecting the cross-section arcs with the area property polygons and marking them in the cross section database.

Cross Section Database

When extracting the cross sections you will be prompted for the name of a cross-section database file. SMS stores all of the cross-section information in a text database file. The cross section database can also be edited independently using the *Cross Section Editor* tools. Extracting cross sections with feature arcs is only way to generate cross-section information, they also can be imported from spreadsheet files (cut and paste), or entered manually.

Related Topics

- 1D River Module Tools

Interpolate Cross Sections

Using the Interpolate Cross Sections command in the River Tools menu you can create any number of additional cross sections between two selected cross sections. This allows you to provide more detail to the hydraulic model. The interpolated cross sections are derived from the geometries of the selected cross-sections. The stationing of the new cross sections are established by evenly distributing the length between the selected cross sections.

New Feature Arcs are created for the interpolated cross sections.

Related Topics

- 1D River Module Tools

Managing Cross Sections

For the new 1D Hydraulic Cross Section coverage, the cross section geometry is stored in text database file on disk. When extracting cross sections they are saved to a new (or existing) database file. This database was the basis for the development of the cross section data in the ArcHydro data model. Cross sections in the database can be used for the development of hydraulic models such as HEC-RAS.

The *Manage Cross Sections* command allows you to create a new database or open an existing database to add geometries, edit existing ones, and provide proper geo-referencing information. It is also possible to open a cross section database using the *Open* command from the *File* menu.

Cross Section Database Definition

When setting up a new database the following attributes can be defined:

- Topo ID – a topographic identifier and description that identifies where the cross section database was derived from. You should create a new Topo ID for each database.
- Line Prop Types – By default WMS uses only a Material ID, but other properties could be defined for general use (they will not immediately be used by supported hydraulic models).
- Point Prop Types – By default WMS uses thalweg, left bank, and right bank but other point properties could be defined for general use.

The cross section database management dialog also allows you to create a new cross section, edit/copy/delete an existing cross section, insert an entire database (merge databases together), convert a cross section database to a coverage (of course the georeferencing of cross sections must be provided for the cross section to be included in the coverage), create a digital terrain model from the cross section geometry, and converting the coverage to line properties.

Related Topics

- 1D River Module Tools

Recompute All Stations

This function can be used if an arc's "Computational Length" value on a river changes. It sets the Start and End station values for each of the arcs with the same river name based on the computational lengths of each arc. So, for example, if you had 3 arcs in a river, the most downstream arc being 30 feet, the next arc being 40 feet, and the next arc being 50 feet, the starting and ending stations would be as follows for each of the arcs:

Arc	Start Station	End Station
1 (downstream)	0.0	30.0
2	30.0	70.0
3	70.0	120.0

These start and end stations are set when you assign a river name to an arc, so you should really never need the "recompute all stations" command unless you change the computational length, start station, or end station values for a centerline arc.

Related Topics

- 1D River Module Tools

RivHyd Data Export

The time steps to export are chosen in the left side of the dialog. Generally the original point locations for the cross-sections and reaches is sufficient, but if more dense information is desired the spacing can be set using the controls on the right side of the dialog. The data will be linearly interpolated along the cross-section or reach between data points. The filename can be chosen at the bottom of the dialog. In order to read the data into SMS as a scatter point file, the extension should be pts.

Back to 1D River Module

RivHyd Post-Processing

After a simulation has been successfully run the solution file contains data at the cross sections. This data normally includes velocity and water surface elevation. Depending on the analysis program used, other data may also be available. The solution data may vary across the cross section. These solution files may be read into SMS by using *File* | *Open*.

Datasets

Data read in from a solution file and associated with the 1D Hydraulic Module are stored as datasets with values at each cross section. In some cases, a cross section may have multiple values such as varied velocities in overbank and channel regions. These datasets can be managed through the *Project Explorer*.

Solution data at various cross-section and reach points for the active dataset may be exported to a text file by selecting *Data* | *Export Solution Data*. This will bring up the River Hydraulic *Solution Export* dialog box.

2D Plots

In order to see the model results, 2D plots must be generated in a plot window. (Future releases of SMS will support the mapping of 1D results onto a Triangulated Irregular Network representing the 1D simulation). The two types of plots that can be created are:

- Cross-section plots which show model results for one or more cross-sections.
- Profile plots which displays model results along a reach. Values for the profile plot are the minimum, average, and/or maximum of the results of individual cross-sections along the reach.

Both plots are created through the *Plot Wizard*.

Back to 1D River Module

3.3. 3D Cartesian Grid Module

3D Cartesian Grid Module

This module is currently being developed for future interfaces.

Related Topics

- 3D Cartesian Grid Display Options
- SMS Modules

3D Cartesian Grid Module Display Options

The properties of the 3D Cartesian Grid data SMS displays on the screen can be controlled through the Display Options dialog. The entities associated with the 3D Cartesian Grid module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available 3D Cartesian Grid display options include the following:

Model specific options

Each model may include model specific display options.

Related Topics

- 3D Cartesian Grid Module
- Display Options

3.4. Cartesian Grid Module

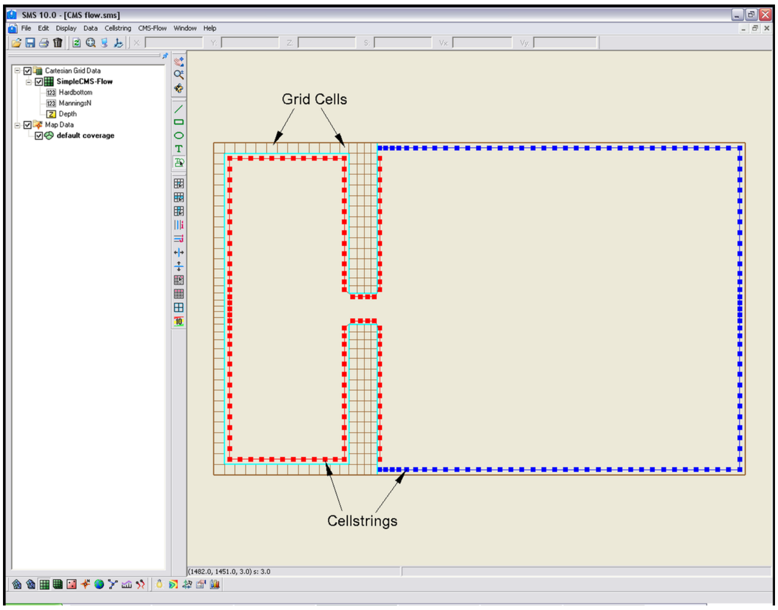
Cartesian Grid Module

At a glance

- Used to create, edit, and visualize rectilinear grids
- Datasets can have values at cells, corners, and midsides

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width.



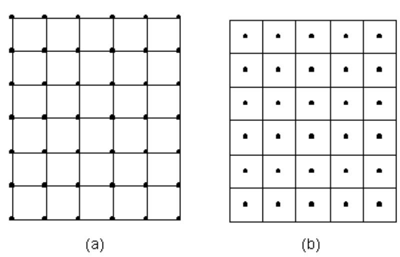
Cartesian Grid Example

It is strongly recommended that grids be created through the Map Module. The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

Grid Types

Multiple types of grids are supported in the 2D Grid module. Generally, a grid can be classified as either a mesh-centered grid or a cell-centered grids. With a mesh-centered grid, the data values are stored at the corners of the grid cells. The BOUSS-2D model is an example of this type. With a cell-centered grid, data values are stored at the cell centers. Both STWAVE and CMS-Wave are example of this type. In additions, a model may generate data at the faces of the cells. In the most complicated cases, a model will use and/or output data in multiple locations. CMS-Flow is an example of this type.



Types of 2D Grids Supported by models SMS. (a) Mesh-Centered Grid (b) Cell-Centered Grid.

When a dataset is imported to a cell-centered grid, there is one value in the dataset for each cell.

In Cartesian grids, row and column boundaries are straight. Each cell center or grid node can have a unique elevation. The grid can also be rotated about the Z axis if desired.

Creating and Editing 2D Grids

Creating 2D Grids

The two main techniques used to create 2D grids are: the ***Create Grid*** command and the *Map → 2D Grid* command.

Create Grid

A new grid can be created by selecting the ***Create Grid*** tool from the *Cartesian Grid Tools*. With this tool active, the user can create a grid by clicking on three points in the graphics window. The first click defines the origin of the grid, the second click defines the orientation of the grid and length of the *I* axis and the third click defines the length of the length of the *J* axis. Once the user clicks three times defining the three points, the *Map → 2D Grid* dialog appears. The values include:

- **Grid Geometry –** This section allows the user to specify the origin, orientation and size of the grid. The fields of these quantities are populated with default values based on the three points. The orientation is measured as an angle from the positive X axis.
- **Cell Options –** This section allows the user to specify the number of cells in each direction in the grid. Several options are available. The user can specify sizes in the I (Delta U)and J (Delta V) directions or a number of columns and rows. If the *Use Grid Frame Size* toggle is checked, the grid will exactly match the dimensions specified in the *Grid Geometry* section. If that option is not checked, the last row and column may extend beyond the specified lengths. This allows the user to specify exact grid size, or exact cell size.
- **Depth Options –** The elevations or depths assigned to each cell or node can be specified as a single value, or the user may select a dataset to interpolate from.
- **Current –** For models that support currents, the user may also specify if current field either as a constant, or interpolated from a vector dataset.

The type and orientation of the grid generated is controlled by the current Cartesian Grid Model.

For some models, specific grid helps are available via a button at the bottom of the dialog.

Map → 2D Grid

The *Map → 2D Grid* command is used to construct a 2D grid using a grid frame feature object in a the current coverage. When the ***Map → 2D Grid*** command is selected, the dialog appears as described above.

If one or more refine points are defined in the conceptual model, the number of rows and columns in the grid will be automatically determined when the grid is created. Thus, these fields cannot be edited by the user and will be dimmed. If refine points are not defined, the user must enter the number of rows and columns.

Editing 2D Grids

Each of the cells in a 2D grid can be active (water) or inactive (land). An inactive cell is ignored when contours or vectors are displayed on the grid and by the numeric engine during computation. If a cell has the potential of becoming active (due to wetting/drying or a similar process), it should be classified as active. Cells status is specified by selecting the cell and assigning a status through the model menu.

Rows and columns can be added to an existing grid that supports variable row/column size by using the **Insert Row**, **Insert Coumn**, **Drag Row**, or **Drag Coumn** tool. (See 2D Grid Tool Palette)

Smoothing 2D Grids

It may be useful to smooth the spatial data stored on a 2D grid for a number of reasons. These reasons include:

- In order to conserve the amount of disk spaced required to store a DEM, many DEM formats store elevations rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat."
- Surveys may include anomalies. Smoothing algorithms blend these bad data points into the surrounding values.
- Datasets may include spurious noise either from physical conditions such as waves or numerical filtering. Smoothing can dampen these variations.

When you right click on the grid in the *Project Explorer*, operations for the grid appear in a pop up window. One of these is the smooth operation.

Converting 2D Grids

2D Grids may be converted to other types of data used in SMS, such as a Scattered Dataset of 2D mesh. 2D Grids can be converted by right clicking on the grid in the *Project Explorer*.

Cartesian Grid Module Tools

See Cartesian Grid Module Tools for more information.

Cartesian Grid Module Menus

See Cartesian Grid Module Menus for more information.

Related Topics

- Cartesian Grid Display Options
- Spectral Energy
- Cartesian Grid Find Cell
- SMS Modules

Cartesian Grid Module Overview

Cartesian grid module

Overview

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width. It is used to create, edit, and visualize rectilinear grids. Datasets can have values at cells, corners, and midsides.

Functionalities

The 2D Cartesian Grid Module contains tools used to construct 2D Cartesian finite difference grids. These grids consist of cells aligned with a rectilinear coordinate system.

Some models limit the grid to be defined with square cells, others limit to constant sized rectangular cells, while others add the flexibility of having variable sizes to the cells (variable row height or column width.

It is strongly recommended that grids be created through the Map Module. The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

Grid Types

Types of 2D Grids Supported by models SMS. (a) Mesh-Centered Grid (b) Cell-Centered Grid. In Cartesian grids, row and column boundaries are straight. Each cell center or grid node can have a unique elevation. The grid can also be rotated about the Z axis if desired.

Creating and Editing 2D Grids

Create Grid. A new grid can be created by selecting the **Create Grid** tool from the Cartesian Grid Tools. With this tool active, the user can create a grid by clicking on three points in the graphics window. The first click defines the origin of the grid, the second click defines the orientation of the grid and length of the I axis and the third click defines the length of the length of the J axis. Once the user clicks three times defining the three points, the Map → 2D dialog appears.

Editing 2D Grids. Each of the cells in a 2D grid can be active (water) or inactive (land). An inactive cell is ignored when contours or vectors are displayed on the grid and by the numeric engine during computation. If a cell has the potential of becoming active (due to wetting/drying or a similar process), it should be classified as active. Cells status is specified by selecting the cell and assigning a status through the model menu.

Rows and columns can be added to an existing grid that supports variable row/column size by using the **Insert Row**, **Insert Column**, **Drag Row**, or **Drag Column** tool. (See 2D Grid Tool Palette)

Smoothing 2D Grids

It may be useful to smooth the spatial data stored on a 2D grid for a number of reasons. These reasons include:

- In order to conserve the amount of disk spaced required to store a DEM, many DEM formats store elevations rounded to the nearest integer value. This causes elevation changes to occur in discrete steps rather than smoothly, as would be the case in nature. In regions of low relief, rounded elevations can cause an area to be artificially "flat."
- Surveys may include anomalies. Smoothing algorithms blend these bad data points into the surrounding values.
- Datasets may include spurious noise either from physical conditions such as waves or numerical filtering. Smoothing can dampen these variations.

When you right click on the grid in the Project Explorer, operations for the grid appear in a pop up window. One of these is the smooth operation.

Converting 2D Grids

2D Grids may be converted to other types of data used in SMS, such as a Scattered dataset of 2D mesh. 2D Grids can be converted by right clicking on the grid in the Project Explorer.

Project Explorer

Cartesian Grid Module Root Folder Right Click Menus

Right clicking on the Cartesian Grid module root folder in the project explorer invokes an options menu with the following options:

- Display Options

Cartesian Grid Item Right Click Menus

Right clicking on a Cartesian Grid item in the Project Explorer invokes an options menu with the following module specific options:

- Smooth – Opens the Cartesian Grid Smoothing Options Dialog.

Model Specific Right Click Menus

Command	Description	Applicable Models
Create Transformed Grid	Opens the Create Transformed Grid dialog. Creates a copy of the grid with a rotated origin. Used to change the I direction for wave models.	CMS-Wave, STWAVE

Menus

The Cartesian Grid Module Data Menu commands include:

Model Specific Menus

- BOUSS-2D
- CMS-Flow
- CMS-Wave
- FATE
- STWAVE
- TUFLOW

Model Commands

- Steering Module
- Switch Current Model

Dataset Commands

- Data Calculator
- Dataset Toolbox
- Create Datasets
- Map Elevation
- Zonal Classification

Visualization Commands / Options

- Contour Options
- Vector Options
- Film Loop

Data Conversion Commands

- Grid → Scatterpoint
- Grid → Map
- Grid → Mesh

Grid Commands

- Find Cell

How do I?

To learn more about how to use the Cartesian Grid Module go to the Tutorials section.

Cartesian Grid Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Cartesian Grid module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Cartesian Grid module tool palette. Depending on the current model, and the type of grids it supports, some of these tools may not be available.

Tool	Tool Name	Description	Right Click Menu

	Select Cell	The Select Cell tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked. When a single cell is selected, its Z coordinate is shown in the Edit Window. The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the Edit Window shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points. With one cell selected, the Edit Window shows the point i,j location. With multiple cells selected, the Edit Window shows the number of selected cells. The number and size of the cells can be changed in the Model Control.	When one or more Cartesian grid cells are selected, and there is a scatter set in the project, a right-click in the graphic window will bring up a menu. One of the options is "Interpolate Bathymetry...". This option brings up the Interpolation dialog where the desired source scatter dataset can be selected. When OK is clicked, SMS will interpolate the selected cell(s)' elevations based on the chosen scatter dataset.
	Select Row	The Select Row tool is used to select cell rows. Rows are selected in the same manner as selecting individual cells.	N/A
	Select Column	The Select Column tool is used to select cell columns. Columns are selected in the same manner as selecting individual cells.	N/A
	Split Grid Column	Insert a new column into an existing grid. This tool splits an existing column into two columns at the location selected by the user.	N/A
	Split Grid Row	Insert a new row into an existing grid. This tool splits an existing row into two rows at the location selected by the user.	N/A
	Drag Column Boundary	Edit column boundary. This tool makes one column narrower while making its neighbor wider.	N/A
	Drag Row Boundary	Edit row boundary. This tool makes one row taller while making its neighbor shorter.	N/A
	Select Cell String	Select a “Cell String”. Allows assignment of boundary conditions.	N/A
	Create Cell String	Create a “Cell String”. This tools allows the user to define a string of cells for later assignment of boundary conditions or flux observations.	N/A

Related Coverages

The grid module currently includes interfaces for:

- BOUSS-2D – phase resolving Boussinesq wave energy and circulation model
- CMS-Flow – hydrodynamic circulation specifically adapted for coastal zone
- CMS-Wave – wave energy model
- STWAVE – wave energy model
- TUFLOW – Coastal, Riverine, and Urban hydrodynamic model with emphasis in flooding applications

2D Grid Files

Two-dimensional grids are stored in 2D grid files. The grids can be either cell-centered or mesh-centered. If the grid is mesh-centered, a set of material IDs may be included in the file. The 2D grid file format is shown in Figure 1, and a sample file in Figure 2.

GRID2D	/* File type */
TYPE i	/* Type of grid. Mesh or Cell centered. */
IJ ±idir ±jdir	/* Card for defining rows, columns. */
DIM nx ny	/* # of cell boundaries in each direction. */
x1	/* X coord. of cell boundaries. */
x2	
.	
.	
x_{nx}	
y1	/* Y coord. of cell boundaries. */
y2	
.	
.	
y_{ny}	
DELEV el	/* Default elevation for grid. */

Figure 1. 2D Grid File Format.

GRID2D
ID 5758
TYPE 1
DELEV 0.000000000000000e+00
IJ -y +x
DIM 4 4
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02
0.000000000000000e+00
3.333333333333334e+01
6.666666666666667e+01
1.000000000000000e+02

Figure 2. Sample 2D Grid File.

2D Grid Files Card Types

The card types used in the 2D grid file format are as follows:

Card Type	GRID2D
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	TYPE		
Description	Defines the type of grid as either cell- or mesh-centered.		
Required	YES		
Format	TYPE i		
Sample	TYPE 0		
Field	Variable	Value	Description
1	i	0,1	The type code: • i = 0 for mesh-centered • i = 1 for cell-centered

Card Type	IJ		
Description	Defines the orientation of the i,j indices.		
Required	YES		
Format	IJ ±idir ±jdir		
Sample	IJ +x -y		
Field	Variable	Value	Description
1	±idir	±x,±y	The direction corresponding to an increasing i index.
2	±jdir	±x,±y	The direction corresponding to an increasing j index.

Card Type	DIM
Description	Defines the dimensions of the grid.
Required	YES
Format	DIM nx ny x1 x2 . . xnx y1 y2 . . yny

Sample	DIM 4 6 0.0 1.0 2.0 4.0 10.0 12.0 14.0 16.0 18.0 20.0		
Field	**Variable**	**Value**	**Description**
1	nx	+	The number of cell boundaries in the x direction.
2	ny	+	The number of cell boundaries in the y direction.
3 to (nx+2)	x1-xnx	±	The coordinates of the x boundaries.
(nx+3) to (nx+ny+2)	y1-yny	±	The coordinates of the y boundaries.

Card Type	**DELEV**		
Description	Defines a default elevation for the grid.		
Required	NO		
Format	DELEV el		
Sample	DELEV 100.0		
Field	**Variable**	**Value**	**Description**
1	el	±	The default elevation.

Back to XMS

Cartesian Grid Coordinates

A projection can now be associated with a Cartesian grid. The data for the grid will be stored in this projection; however, the grid can still be displayed in any projection chosen by the user. When the SMS project's projection ("working projection") is changed, the grid will be converted "on the fly." While the display will be changed, the data will remain in the original projection. This method will reduce rounding errors in the data introduced when converting coordinates.

Editing the Grid

When the grid is displayed in a projection different than its own, it will not be editable. The "working projection" must match that of the grid to be able to edit. The grid's right-click command, "Work in grid projection", will set the "working projection" to the grid's projection.

Changing the Grid Projection

When a grid is created, the projection is defaulted to the "working projection". The grid's projection can be changed using the "Projection..." and "Reproject..." commands in the grid's right-click menu.

Floating Projection

If a grid is read in from a file that does not specify a projection, the grid will "float" in whichever projection is the working projection. If a grid is floating, the "Projection..." command in the right-click menu will be followed by "floating". To assign a projection to the grid, select the "Projection..." command and select a projection.

Cartesian Grid Data Menu

Most of the SMS modules have a Data Menu, but the items in this menu are different for each module. The Cartesian Grid Module Data Menu commands include:

Model Commands

- Steering Module – Launches the steering tool.
- Switch Current Model – Allows user to change current active model.

Dataset Commands

- Data Calculator – Can be used to perform mathematical operations with datasets.
- Dataset Toolbox – Contains tools for working with datasets.
- Create Datasets – Opens a dialog that can be used to create functions for the entire mesh or active scatter set.
- Map Elevation – Allows use of another functional dataset as the mapped elevation function.
- Zonal Classification – Tool to identify areas that meet as set of criteria.

Visualization Commands / Options

- Contour Options – opens a dialog where dataset specific contour options can be defined.
- Vector Options – opens a dialog where options to generate vectors can be edited.
- Film loop – opens the *Film Loop Setup* wizard.

Data Conversion Commands

- Grid → Scatterpoint
- Grid → Map
- Grid → Mesh

Grid Commands

- Find Cell – Used to locate a cell either with a specific i,j location, or near a specific location.

Related Topics

- Cartesian Grid Module

Cartesian Grid to Map

The Grid → Map command in the data menu (Cartesian grid module) is equivalent to the 2D Grid → Map command in the right click menu on a grid object in the project explorer. It is used to convert attributes of the Cartesian grid into feature objects on a coverage (map module). All generated features are added to the current or active coverage. If a new coverage is desired, it should be created prior to issuing this command.

The command includes the following options:

Land/Water Boundary → Arcs

This option only applies to Cartesian grids which support cell attributes supporting land and water cells (CMS-Flow). When this option is selected, the cell faces between cells of these opposing types are converted to feature arcs. For grid cells that do not support cell attributes, this options functions identically to the Grid Boundary → Arcs option.

Grid Boundary → Arcs

This option generates a feature arc along all cell edges of the boundary of the Cartesian grid.

Observation Cells → Points

This option can be applied to either arc generation option above. It only applies to Cartesian grids that support the cell attribute of observation cells. If this toggle is selected, a feature point is created at the centroid of each cell with the observation point attribute.

This command has minimal usefulness since the feature objects generated are step functions. It is recommended that other data sources, such as a scatter set or a shapefile be used to define these features, but if only the numerical model exists, this command can be used to help construct a conceptual model.

Cartesian Grid to Scatter

The Grid → Scatter command in the data menu (Cartesian grid module) is equivalent to the 2D Grid → 2D Scatter command in the right click menu on a grid object in the project explorer. It is used to convert the grid cell corners into a scattered data set (scatter module).

Each cell corner in the grid is converted to a scatter vertex. SMS computes one data set from the elevation data set of the grid and one data set for each functional data set on the grid. If the Cartesian grid is cell centered (data at the center of the cell), SMS averages the values of the four surrounding cells to compute a value for the scatter set vertex. If the Cartesian grid is a mesh centered grid (BOUSS2D), SMS assigns the value from the cell corner to the scattered vertex.

Each cell in the grid is converted to two triangles in a TIN.

This command allows the visualization of the data on a matching geometric object.

SMS:Cartesian Grid to Mesh

The Grid → Mesh command in the data menu (Cartesian grid module) is equivalent to the 2D Grid → 2D Mesh command in the right click menu on a grid object in the project explorer. It is used to convert the grid cell corners into a mesh or unstructured grid object (mesh module).

Each cell corner in the grid is converted to a mesh node. SMS computes an elevation for the mesh node as the average of the four surrounding cells in the grid for cell centered grids, or the elevation of the corner for mesh centered grids.

Each cell in the grid is converted to two triangular elements.

This command allows the visualization of the data on a matching geometric object.

Cartesian Grid Find Cell

The Find Cell command from the Data menu is used to locate a cell either with a specific i,j location, or near a specific location. When this command is executed the Find Cell dialog opens.

When the *Find by (I,J)* option is selected, the cell with the specified i,j is highlighted in red. If there is no cell with the specified i,j, an error message is given. Conversely, when the *Find by nearest (x,y) coordinates* option is selected, the cell containing the specified coordinate is highlighted with red. If no cell contains the x,y location, an error message is given. With either of these methods, the found cell becomes selected in addition to being highlighted.

Return to Cartesian Grid Module

Cartesian Grid Module Display Options

The properties of all mesh data that SMS displays on the screen can be controlled through the Cartesian Grid tab of the Display Options Dialog. This dialog is opened by right clicking on the Cartesian Grid Data entry in the Project Explorer and selecting the Display Options command. (It can also be accessed from the from the Display menu or the display options Macro.)

Cell display

SMS allows the user to specify the level of quality for displaying cells. This is done at the top of the display options dialog by choosing 1 quad, 4 quad, 4 triangles or 8 triangles as the display option. We have found that 1 quad is the fastest display option, but in situations of high relief, the cell may be distorted from a flat quad and this representation can leave gaps in the display. For such situations, 4 quads generally solves the display issues, but takes a little longer. For highest quality of display, and to assure matching of the cell display with the contours, 8 triangles can be used.

The cells are colored based on their types. Land cells are colored separately from water cells. Special cells are marked with symbols. These could be observation stations (probes), or specially marked cells for another purpose.

Cell String display

Some models support cell strings (M2D, BOUSS2D), at which specific attributes or boundary conditions are specified. Each cell string type has its own display attributes.

Grid display

Other entities associated with the Cartesian Grid module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available mesh display options include the following:

- **Contours –** The contours are drawn for the active scalar data set. All standard contour display options are supported for cartesian grid contours.
- **Vectors –** The cartesian grid vectors are drawn for the active vector dataset. All standard vector display options are supported.
- **Grid Boundary –** A line around the perimeter of the cartesian grid can be drawn. This is useful when the cells are turned off. User specifies line color and thickness.

- **Computational Domain –** It can be useful to delineate the active (or water cells) from the rest of the grid. This option allows for specification of a line to outline the computational cells.
- **Cell id**
- **Cell i, j –** The i,j coordinate can be drawn in each cell.
- **Elevations –** The current scalar value can be drawn in each cell.
- **IJ triad –** Arrows can be displayed at the origin of the grid showing the i and j directions.
- **Display Inactive Grids –** SMS supports the ability to store/manipulate multiple Cartesian grids at the same time. This allows for functionality such as nested grids in STWAVE and steering between STWAVE and M2D. However, only one grid can be edited at a time. This is the "active" grid. The outline of other grids can be displayed using the Display Inactive Grid option.
- **Functional Data Surface**

Model specific options

Each model may include other display options associated only with that model or options slightly modified from those described above. For example, the types of cell strings supported by each model are different. This is reflected by slightly different options for each model. Details on these entities can be found in model specific documentation.

Related Topics

- Cartesian Grid Module
- Display Options

Cartesian Grid Module Right Click Menus

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a Cartesian Grid Module item.

Cartesian Grid Module Root Folder Right Click Menus

Right clicking on the Cartesian Grid module root folder in the project explorer invokes an options menu with the following options:

- Display Options

Cartesian Grid Item Right Click Menus

Right clicking on a Cartesian Grid item in the Project Explorer invokes an options menu with the following module specific options:

- **Smooth –** Opens the Cartesian Grid Smoothing Options Dialog.

Model Specific Right Click Menus

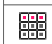
Command	Description	Applicable Models
Create Transformed Grid	Opens the Create Transformed Grid dialog. Creates a copy of the grid with a rotated origin. Used to change the I direction for wave models.	CMS-Wave, STWAVE

Related Topics

- Project Explorer Right Click Menus
- Menu Bar

Cartesian Grid Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Cartesian Grid module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Cartesian Grid module tool palette. Depending on the current model, and the type of grids it supports, some of these tools may not be available.

Tool	Tool Name	Description	Right Click Menu
	Select Cell	The Select Cell tool is used to select a grid cell. A single cell is selected by clicking on it. A second cell can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple cells can be selected at once by dragging a box around them. A selected cell can be de-selected by holding the *SHIFT* key as it is clicked. When a single cell is selected, its Z coordinate is shown in the Edit Window. The Z coordinates can be changed by typing in the edit field, which updates the depth function. If multiple cells are selected, the Z Coordinate field in the Edit Window shows the average depth of all selected cells. If this value is changed, the new value will be assigned to all selected points. With one cell selected, the Edit Window shows the point i,j location. With multiple cells selected, the Edit Window shows the number of selected cells. The number and size of the cells can be changed in the Model Control.	When one or more Cartesian grid cells are selected, and there is a scatter set in the project, a right-click in the graphic window will bring up a menu. One of the options is "Interpolate Bathymetry...". This option brings up the Interpolation dialog where the desired source scatter dataset can be selected. When OK is clicked, SMS will interpolate the selected cell(s)' elevations based on the chosen scatter dataset.
	Select Row	The Select Row tool is used to select cell rows. Rows are selected in the same manner as selecting individual cells.	N/A
	Select Column	The Select Column tool is used to select cell columns. Columns are selected in the same manner as selecting individual cells.	N/A
	Split Grid Column	Insert a new column into an existing grid. This tool splits an existing column into two columns at the location selected by the user.	N/A
	Split Grid Row	Insert a new row into an existing grid. This tool splits an existing row into two rows at the location selected by the user.	N/A
	Drag Column Boundary	Edit column boundary. This tool makes one column narrower while making its neighbor wider.	N/A
	Drag Row Boundary	Edit row boundary. This tool makes one row taller while making its neighbor shorter.	N/A
	Select Cell String	Select a “Cell String”. Allows assignment of boundary conditions.	N/A

	Create Cell String	Create a “Cell String”. This tools allows the user to define a string of cells for later assignment of boundary conditions or flux observations.	N/A

Related Topics

- SMS:Cartesian Grid Module

Grid Frame

For a Cartesian Grid coverage in the Map module, the *Grid Frame* menu item brings up the *Grid Frame* dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. The grid frame sets the bounds of the Cartesian grid that will be created when the user selects the *Map → 2D Grid* command.

Return to Grid Frame Dialog

Grid Smoothing

The Cartesian Grid Smoothing Options Dialog is opened by right clicking on a Cartesian Grid item in the Project Explorer. The following options can be specified:

- **Filter size –** This determines how many neighbors are included when smoothing the grid. Options are 3x3 and 5x5.
- **Number of iterations –** This specifies how many passes should be made with the smoothing algorithm.
- **Max. elevation change –** This value specifies the maximum allowable elevation change per iteration for each cell.
- **Filter ratio –** The new cell elevation is computed using the original elevation (at the beginning of the iteration not the whole process) and the "blurred" elevation. The filter ratio defines how far the elevation is changed between the original elevation and the "blurred" elevation. A filter ratio of 1.0 would replace the existing elevation with the "blurred" elevation. A filter ratio of 0.0 would be pointless as it wouldn't change the elevations. A filter ratio of 0.5 would give a new elevation that is the average of the original elevation and the blurred elevation.
- **Filter Range –** The start and end index values specify the extents of the smoothing. Defaults to the grid extents.
  - **Column start –** The column index on which to start the smoothing process.
  - **Column end –** The column index on which to end the smoothing process.
  - **Row start –** The row index on which to start the smoothing process.
  - **Row end –** The row index on which to end the smoothing process.
- **Only modify selected cells/cell locations –** If this option is selected, only the cells or cell locations (if model uses elevations at centers, faces, and corners) that are selected are smoothed. Cells or cell locations not selected may be used to compute "blurred" elevations but their elevations are never modified.

Related Topics

- Cartesian Grid Module Right Click Menus
- Cartesian Grid Module

Refine Point Dialog

The Refine Point dialog is used to set the attributes for a refine point represented by a feature point in a Cartesian Grid model coverage. Attributes that can be specified for each refine point include:

- Refine grid in I direction(checked = on)
  - Base cell size – Specify the cell I size in the vicinity of the refine point.
- Refine grid in J direction(checked = on)
  - Base cell size – Specify the cell J size in the vicinity of the refine point.

Only refine points located within a grid frame are used when the Map → 2D Grid command is executed. Refine points are not available for all models, since some Cartesian Grid models require uniform cell sizes.

Related Topics

- Feature Objects Menu

3.5. Curvilinear Grid Module

Curvilinear Grid Module

The curvilinear grid module contains tools used to work with curvilinear grid data. Curvilinear grids consist of nodes that are grouped together to form cells. These nodes and cells define the computational domain of the numerical model. In addition to nodes and cells, a curvilinear grid may store additional information such as material values assigned to elements and boundary conditions assigned to nodes. In general, this additional information is used as input data for the numerical model.

Nodes

Nodes are the basic building blocks of cells in a curvilinear grid. Nodes store elevation and other dataset values. Nodes can also be used for building nodestrings and assigning boundary conditions. The density of nodes helps determine the quality of solution data and can be important to model stability.

Cells

Cells are used to describe the area to be modeled. Cells are formed by joining exactly four nodes. No more than four cells may join at a single node. If four cells join at a single node, the node cannot be a boundary node. Cells are identified by a unique i, j index.

Nodestrings

A collection of nodes can be formed into a nodestring. Nodestrings are most commonly used to assign boundary conditions such as a flowrate or water-surface elevation. Nodestrings can also be used for mesh renumbering, forcing break lines, and boundary smoothing. Finally, a nodestring can store attributes pertinent to a location such as the total flow nodestring.

Curvilinear Grid Models

The curvilinear grid module currently includes interfaces for:

- LTFATE

Curvilinear Grid Module Tools

See Curvilinear Grid Module Tools for more information.

Curvilinear Grid Module Menus

See Curvilinear Grid Module Menus for more information.

Curvilinear Grid Display Options

See Curvilinear Grid Display Options for more information.

Related Topics

- Curvilinear Grid Display Options
- SMS Modules

Curvilinear Grid Display Options

Curvilinear Grid Module Display Options

The properties of the curvilinear grid data SMS displays on the screen can be controlled through the *Display Options* dialog. The entities associated with the curvilinear grid module with display options are shown below. Some of these entities also have an associated *Options* button. For these entities, additional display options are available. The available curvilinear grid display options include the following:

- Nodes - A circle is filled around each node. The user can specify the radius and color of these circles. The Options button is used to set the display of nodal boundary condition data. The dialog that opens when this button is clicked depends on the current numerical model.
- **Edges** – Cell edges.
- **Contours** – Contours are drawn for the active scalar dataset. Use the contours tab to change contour options.
- **Boundary** – A line is drawn around the perimeter of the curvilinear grid.
- **Element Fill** – Elements can be filled using the following options:
  - **None**
  - **Materials** – Elements are filled using the material assigned to the element.
  - **Mesh quality** – Elements are filled using a user specified mesh quality metric. For a description of the mesh quality metrics, please see the VERDICT Manual [1] which contains the mathematical definition of each quality metric. The VERDICT website contains further information on the VERDICT [2] mesh verification code library.
  - **Solid color** – Elements are filled using a solid color.
  - **Texture mapping** – An image is draped over the mesh elements.

Model specific options

Each model may include model specific display options. These appear at the bottom of the display options dialog.

Related Topics

- Curvilinear Grid Module
- Display Options

References

[1] http://www.vtk.org/Wiki/images/6/6b/VerdictManual-revA.pdf
[2] http://cubit.sandia.gov/verdict.html

Curvilinear Grid Module Menus

When one or more active nodestrings have been created, and the "Select Nodestrings" tool is selected, a set of menu's becomes available by right clicking on the mouse. The menu items operate on the active nodestrings:

- **Delete Selected –** deletes the selected nodestrings.
- **Merge Selected –** will merge two or more selected nodestrings to form a single nodestring. Nodestrings must share the same endpoints to be merged.
- **Clear Selections –** unselect all of the selected nodestrings.
- **Select All –** selects all nodestrings.

Related Topics

- Curvilinear Grid Module

Curvilinear Grid Module Tools

The following tools are active in the dynamic portion of the *Tool Palette* whenever the Curvilinear Grid Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the *Graphics Window* with the cursor depends on the current tool. The table below describes the tools in the Curvilinear Grid tool palette.

<table>
<tr><th>Tool</th><th>Tool Name</th><th>Description</th><th>Right Click Menu</th></tr>
<tr><td></td><td>Select Curvilinear Grid Node</td><td>The Curvilinear Grid Node tool is used to manually selected an individual node. A second node can be added to the selection list by holding the SHIFT key while selecting it (Not implemented yet). Multiple nodes can be selected at once by dragging a box around them. A selected node can be deselected by holding the SHIFT key as it is clicked.</td><td>N/A</td></tr>
<tr><td></td><td>Select Element</td><td>The Select Element tool is used to select the element. A single element is selected by clicking on it. A second element can be added to the selection list by holding the SHIFT key while selecting it. Multiple elements can be selected at once by dragging a box around them. A selected element can be deselected by holding the SHIFT key as it is clicked.</td><td>N/A</td></tr>
<tr><td></td><td>Create Nodestrings</td><td>The Create Nodestrings tool is used to create node string. Nodestrings are used for operations such as assigning boundary conditions, forcing breaklines into the mesh, and renumbering the mesh. To create a nodestring:
1. Click on a node. The node will be highlighted in red and a prompt will be shown in the Help Window.
2. Click on any node to add it to the nodestring. The selected node is also highlighted in red and a solid red line is drawn between the two nodes. Continue adding nodes to the nodestring in this manner.
  1. Note: For most operations, nodes in the nodestring should be adjacent, but this is not required.
  2. Press the BACKSPACE key to backup one node. Press the ESC key to abort the nodestring creation.
  3. Double-click a node or press the ENTER key to end the nodestring creation.
3. The SHIFT and CTRL keys assist in creating large nodestrings which are made up of adjacent nodes. These can be used after at least one node has been selected and function as follows:
  1. SHIFT. Holding down the SHIFT key and selecting another node will add to the nodestring all nodes between the two. The path chosen is the shortest distance between the two nodes. This is useful for creating continuity strings which run along a cross section of the mesh.
  2. CTRL. Holding down the CTRL key and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going counter clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.
  3. CTRL + SHIFT. Holding down both the CTRL and SHIFT keys and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.</td><td>N/A</td></tr>
<tr><td></td><td>Select Nodestrings</td><td>The Select Nodestrings tool is used to select nodestrings. A nodestring is selected by clicking on it or by dragging a box around it. A second nodestring can be added to the selection list by holding the SHIFT key while selecting it. Multiple nodestrings can be selected by dragging a box around their icons. A selected nodestring can be deselected by holding the SHIFT key as its icon is clicked. The Edit Window shows the number of nodestrings selected.</td><td>See the right click menu options for each module.</td></tr>
</table>

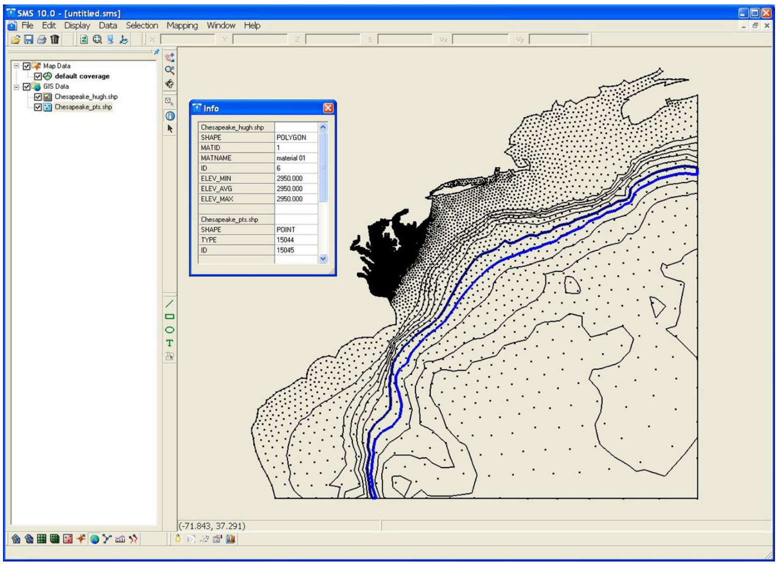
Related Topics

- Curvilinear Grid Module

3.6. GIS Module

GIS Module

At a glance


GIS Data in SMS

- Open and visualize GIS data
- Supports ESRI and MapInfo formats
- Uses Mapobjects for ESRI files if available to use ArcGIS visualization options
- GIS data can be converted to feature data (map module)

The GIS module has been separated from the Map module in order to define a more integrated and separate approach to linking with GIS data. The GIS module has two separate modes, although the primary functions are available in either mode. The main reason that the GIS data has been separated from the map module is to allow users to handle large files more efficiently when creating hydrologic models. For example the GIS module allows you to import large files and then select and convert to feature objects only the portions that are needed. The conversion to feature objects is both time consuming and potentially memory intensive, and so managing your GIS data in this fashion is more efficient (the equivalent GIS functionality is to clip out just the parts of the data you need for import).

Some of the key functionality available in either mode includes:

- Efficient management of large datasets
- Graphical selection of features
- Mapping of selected features to feature objects in map coverages
- Viewing attribute tables
- Joining additional attribute tables based on a key field (i.e. joining the hydrologic soils group attribute to a STATSGO/SSURGO shapefile).

Using the GIS Module with a License of ArcView®

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

To use SMS with ArcView®, do the following:

- Activate the GIS Module.
- Enable ArcObjects by selecting Data | Enable ArcObjects.
- Open the desired shapefile by selecting Data | Add Shapefile Data... and browse for the file. The file should now appear in the Project Explorer.
- Right-click on the imported shapefile and select Properties. The ArcGIS Properties window will appear.
- Click on the Symbology tab and the shapefile properties can be edited.

Using the GIS Module without a License of ArcView®

Most of the same functionality that exists for users with licenses of ArcView® is available to users without. The primary differences are that you can only read layers that are in the shapefile format, and all of the display and symbology available with ArcView® is not available. Points, lines, and polygons are displayed in a single color and not filled. Further some of the queries for selection are not supported without a license to ArcView®.

GIS Module Tools

See GIS Module Tools for more information.

GIS Module Menus

See GIS Module Menus for more information.

Related Topics

- Map Module
- ArcObjects
- Shapefiles
- Modules

Importing Shapefiles

ARC/INFO or ArcView® shape files provide an easy method to import GIS data into SMS. Unfortunately the shape file format is extremely redundant, meaning that points or lines that are shared by lines or polygons are multiply defined.

Therefore, in order for you to convert a shape file to a SMS coverage it may take up to several minutes (depending on size) to build the correct line or polygon topology. This was very problematic in previous versions because SMS often bogged down when reading moderately large files. This is one of the primary reasons that the new GIS module has been developed and with or without a license to ArcObjects shapefile data can now be managed better by SMS.

With the addition of the GIS module there are now two different ways to import shapefile data.

Direct Conversion of Shapefile Data to Coverages

The first is the traditional method which allows you to load a shapefile layer directly into a coverage.

You can then map attribute fields of the shapefiles database (*.dbf) file to their pertinent SMS parameters .

Using the GIS Module to Convert Shapefile Data to Coverages

When opening a shapefile in the GIS module using the Add Shapefile Data or Add Data commands SMS first reads the points/lines/polygons into a simple display list and does not try to "build" topology by connecting arcs at nodes, and eliminating shared edges of polygons as required when creating a coverage. This makes the display and selection of the polygons much easier and more efficient. You can then select only the polygons you wish to convert to a coverage and map them. In this way you will only be building topology for the selected polygons.

Cleaning Imported Shapefile Data

If you intend to use the data from the shape file in more than one session you should save it as a SMS map file after importing/mapping the first time. Further, after importing the shape files you may wish to consider the following:

1. Clean the feature objects in order to snap nodes within a certain distance, intersect arcs, and eliminate dangling arcs.
2. Build Polygon so that SMS can define the appropriate conceptual model. After intersection of arcs, reordering of streams, etc. it is often necessary to rebuild the polygon topology so that the topologic structure is consistent.

Return to Shapefiles

GIS Module Tools

The following tools are active in the dynamic portion of the Tool Palette whenever the GIS module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window with the cursor depends on the current tool. The table below describes the tools in the GIS tool palette.

Tool	Tool Name	Description
	Select ArcObject	The Select ArcObject tool is used to select ArcObjects in the Graphics Window. This tool is only available if ArcView is installed locally on the user's desktop.
	Get ArcObject Attributes	The Get ArcObject Attributes tool is used to select GIS objects and display information relating to that object in an Identify Results dialog. This tool is only available if ArcView is installed locally on the user's desktop.
	Select ArcObject	The Select ArcObject tool is used to select shape objects in the Graphics Window.

Related Topics

- GIS Module

GIS Module Menus

The following menus are available in the the GIS Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

Data

Most of the SMS modules have a Data menu, but the items in this menu are different for each module. The GIS Module commands include:

GIS Data Menu Commands

- **Enable ArcObjects**

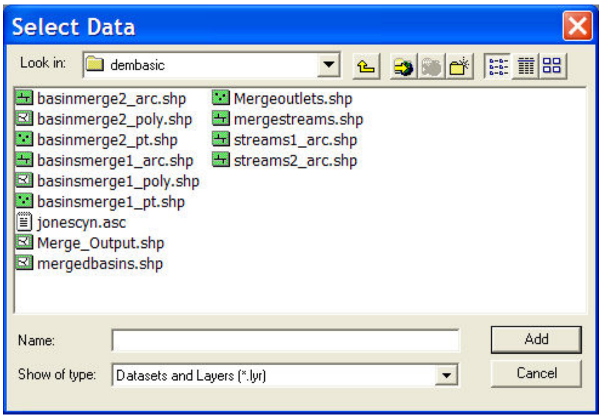
  ArcObjects is a development platform provided by ESRI [1] that allows developers of other applications (such as SMS) to incorporate ArcView/ArcGIS capability directly within their application. SMS can use ArcObjects to access some of the same functionality in SMS that is available in ArcView, providing you are running on a computer that has a current license of ArcView.

  The Data | *Enable ArcObjects* command queries the ESRI license manager for ArcView/ArcGIS to see if a license exists. If a valid license is found then the ArcView functionality within SMS is enabled and you will be allowed access. If a license is not found then the ArcView specific features remain unavailable.

- **Add Data**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

The *Add Data* command is only available if ArcObjects have been enabled. The *Add Data* command is used to open Datasets and layers (*.lyr) files into SMS using ArcView®.



Select Data Dialog

When ArcObjects is enabled you are able to load any of the ESRI supported formats, including shapefiles, coverages, geodatabases, grids, images, CAD files and others, as GIS data layers in SMS. These data can then be converted to feature objects in map coverages.

- **Add Shapefile Data**

The Add *Shapefile Data* command is only available if ArcObjects have **NOT** been enabled. The *Add Shapefile Data* command is used to open a Shapefile (*.shp) into SMS.

- **Add MIF/MID File Data**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

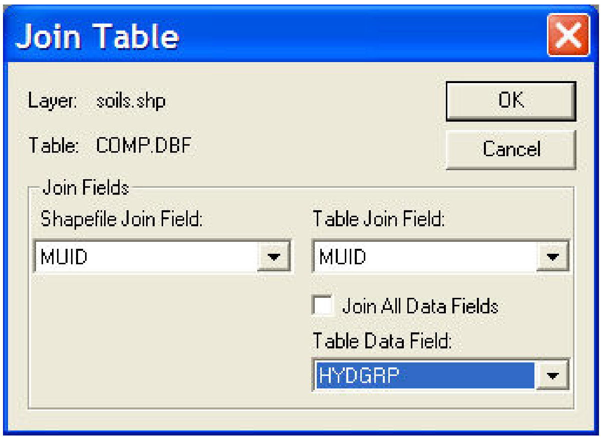
The *Add MIF/MID Data* command is used to open a MIF/MID file (*.mif) into SMS using ArcView®.

- **Attribute Table**

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

The *Attribute Table* command opens the *Attribute Table* dialog. The shapefile layer of interest is specified in the *Attribute Table* dialog. Once the shapefile layer has been specified, the *Attributes* dialog will be shown. The attributes for each record in the specified layer can be viewed in the *Attributes* dialog.

- **Join Table to Layer**



Join Table Dialog

SMS uses ArcObjects to incorporate much of the ArcMap® functionality directly. You can open any ArcView® supported file (coverages, shapefiles, geodatabases, images, CAD, grids, etc.) and use the ArcView® Display Symbology properties to render the GIS data and then display it in SMS.

The Attribute Table command opens the *Attribute Table* dialog. The shapefile layer of interest is specified in the *Attribute Table* dialog. Once the shapefile layer has been specified, the corresponding DBF Table File (*.dbf) must be selected. The *Join Table* dialog is then shown.

The *Join Table to Layer* command, available when right-clicking on a layer in the *Project Explorer*, allows you to join the attributes of one database file (*.dbf) to the features of a GIS layer based on a key attribute field. This is particularly important when the features are stored in a shapefile with a minimal set of attributes, and additional attributes are stored in a separate *.dbf file. The two files are related based on an attribute field named MUID. Other GIS data layers may be similar where the features contain some kind of key indexing field and the attributes are stored in a separate table that can be joined to the features based on the index field values.

After selecting the *Join Table to Layer* command you will be prompted for the database file you wish to join using the standard *select file* dialog. The *Join Table* dialog will then appear and you will be asked to select the *Join Field* from the GIS data layer attributes and the *Join Field* from the table you are joining to the GIS data layer. Often these field names will be the same as in the example below, but they are not required to be the same. The important thing is that they contain similar information from which a join can be made. Finally, you can select to join all of the attributes from the join table or just add a specific field.

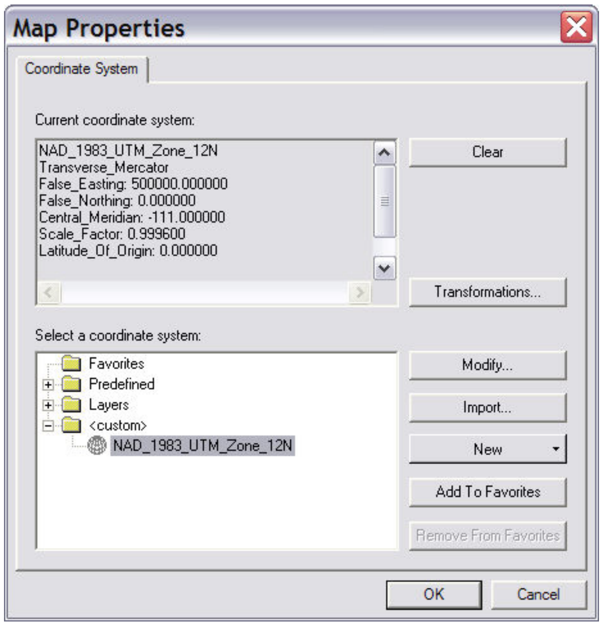
The join does not permanently alter the GIS data layer on the hard drive of your computer.

- **Layer Properties**

The *Layer Properties* command opens the *Select a layer* dialog. The shapefile layer of interest is specified in the *Select a layer* dialog. Once the shapefile layer has been specified, the *Layer Properties* dialog is shown. See the ESRI ArcGIS documentation [2] for further explanation of the *Layer Properties* dialog. The *Layer Properties* command is only available if ArcObjects have been enabled.

- **Map Properties**

The Map Properties dialog is used when ArcObjects® is enabled to specify the coordinate system that you wish to display/map features from the ArcGIS® data layer. An ArcGIS® data layer should have a currently defined coordinate system associated with it. If the coordinate system is geographic (latitude/longitude), then ArcObjects® is able to "guess" correctly at the projection. Using the coordinate system as defined in the *Map Properties* you can specify the coordinate system you wish to use to display features/rasters. While this does not change the actual geometry of the layer, it will display in the main graphics window according to this projection and any data mapped to coverages will be mapped into the coordinate system specified by the *Map Properties*.



Map Properties Dialog

GIS layers can have an associated global projection. If a layer has an associated projection, the entities will be automatically displayed in the current project projection. GIS projection information can come from multiple sources:

1. **Files –** ESRI shapefiles can have an associated prj file that contains the projection information. MapInfo MIF/MID files contain projection within the MIF file.
2. **Assigned in SMS –** You can assign a projection to a layer by right-clicking on the layer and choosing *Coordinate Conversions*. When you do this, SMS will save a PRJ file or a new set of MIF/MID files with the updated information.

Selection

The GIS Module Selection Menu commands are only available if ArcObjects have been enabled. The GIS Module Selection Menu commands include:

GIS Selection Menu Commands

Command	ArcObjects® Required	Description
Select by Attributes	Yes	Opens the ArcObjects® Query Wizard Dialog. See the ESRI ArcGIS documentation [2] for further explanation of the Query Wizard dialog.
Select by Location	Yes	Opens the ArcObjects® Select By Location Dialog. See the ESRI ArcGIS documentation [2] for further explanation of the Select By Location dialog.
Clear Selected Features	Yes	Clears the current selection

Interactive Selection Method	Yes	Change the ArcObjects® selection options in use. Options include: • Create New Selection • Add to Current Selection • Remove from Current Selection • Select from Current Selection
Selectable Layers	Yes	Opens the ArcObjects® Check Selectable Layers dialog. The layers to be selected can be specified.

Mapping

The GIS Module Selection Menu commands are only available if ArcObjects have been enabled. The GIS Module Selection Menu commands include:

GIS Selection Menu Commands

Command	ArcObjects® Required	Description
Arc Objects → Feature Objects	Yes	Opens the GIS to Feature Objects Wizard
Shapes → Feature Objects	No	Opens the GIS to Feature Objects Wizard
Feature Objects → Geodatabase	Yes	Saves the Feature Objects as a Personal Geodatabase file (*.mdb)

References

[1] http://www.esri.com/
[2] http://support.esri.com/

GIS Module Display Options

The properties of the GIS Module data SMS displays on the screen can be controlled through the Display Options dialog. The entities associated with the GIS Module module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available GIS Module display options include the following:

Display Option	Description
Text font	Controls the display of text. The size, color, style, and font of the text can be adjusted.

Related Topics

- GIS Module
- Display Options

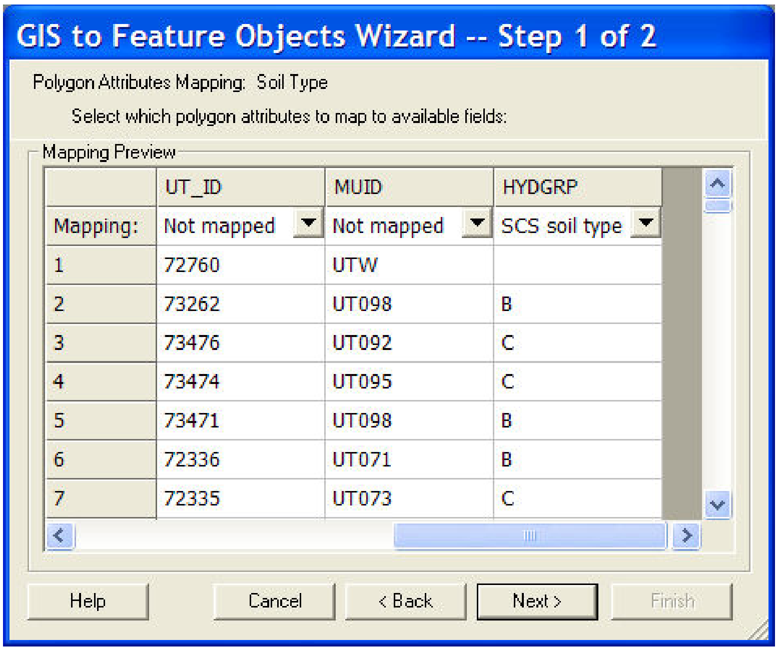
ArcObjects

ArcObjects® is a development platform provided by ESRI® [1] that allows developers of other applications to incorporate ArcView/ArcGIS® capability directly within their application. ArcObjects® is used to incorporate ArcView® functionalities into XMS software. This allows users to use ArcView® functionality within XMS software. In order to use ArcView® functionality, a current license of ArcView® must be installed. Without a license, much of the same functionality is available, the primary differences being that only the shapefile format is supported, and many of the selection and display capabilities are minimal.

Related Topics

- GIS Module (GMS)
- GIS Module (SMS)
- GIS Module (WMS)

GIS to Feature Objects Wizard



GIS to Feature Objects Wizard -- Step 1 of 2

While future versions of the XMS software may be able to process some commands directly from the GIS data layers, currently the user must map all desired features as part of model development to feature objects in a map coverage. One way to do this is to convert an entire shapefile directly to a map coverage. One problem with this approach is that the extents of the GIS data layer may be much larger (i.e. an entire state) than the area of interest. In this case, it may be more efficient to select only those GIS features (points, lines, polygons) that overlay your study area and map those to feature objects in a map coverage.

Selecting the ***Mapping* | *ArcObjects* → *Feature Objects*** command (with an ArcObjects license), or the ***Mapping* | *Shapes* → *Feature Objects*** (without an ArcObjects license) launches a mapping wizard which guides you through the process of converting your GIS data layer features to feature objects in a map coverage. Before beginning the mapping process you should first go to the map module and make sure that the currently active coverage is the coverage you wish to map GIS data layer features to. Also, you will associate the GIS attributes with coverage attributes, so you must make sure the coverage attributes are defined before you can do the conversion.

After making sure you will be mapping to the correct coverage select the GIS features which overlay your study area and that you wish to map (this is done with the selection tool(s) in the GIS module). If you wish to map all the features you can choose the ***Edit* | *Select All*** command, or just execute the Mapping command, and you will be prompted if you want to convert all features since none are selected.

If you have ArcObjects enabled you will see that the ***Mapping* | *ArcObjects* → *Feature Objects*** command is activated whereas if you do not have ArcObjects enabled you will see that the ***Mapping* | *Shapes* → *Feature Objects*** command is activated. After choosing the appropriate mapping command you will see the Mapping Wizard shown below. This wizard will guide you through the rest of the process. The first dialog in the mapping wizard contains instructions and marks the beginning point of mapping for selected features. The first of two steps is to map the GIS attribute fields of the features to the coverage attributes. Common attribute names are automatically mapped.

The second step marks the end of the wizard and after selecting Finish all selected features will be converted to feature objects within the active coverage. Attributes of mapped fields will be saved accordingly as attributes of the feature objects.

Generic Model arc or node mapping

Users can now bring in GIS data (shapefile or MIF/MID) and convert this data to generic model node or arc attributes. First you should have arc and/or node boundary conditions defined in the Generic Model. Once those are defined, you will see them in the drop down box when mapping. Each boundary condition will contain a (on/off) item followed by parameters. Example:

- Hydro→(on/off)
- Hydro→Manning
- Hydro→Flowrate

Mapping "Hydro→(on/off)" will turn on or off the node/arc. Mapping a parameter such as "Hydro->Manning" will automatically assign the node/arc as "on" unless "Hydro→(on/off)" is explicitly mapped to "off".

Related Topics

- GIS Module (GMS)
- GIS Module (SMS)
- GIS Module (WMS)

GIS Module Right Click Commands

The following right click commands are available for GIS objects:

Convert to TUFLOW Rainfall Boundary Conditions

If the format is correct, the GIS data can be converted into TUFLOW rainfall boundary polygons on a TUFLOW 1D/2D BCs and Links coverage. For more information, see TUFLOW Boundary Conditions.

Related Topics

- GIS Module
- GIS Module Tools

3.7. Map Module

Map Module

At a glance

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

- Create and edit GIS like data
- Used to create conceptual models as well as data for other purposes
- Conceptual model is a geometry (mesh/grid) independent representation of the numeric model domain and/or boundary conditions
- Conceptual models can be converted to model geometry and boundary conditions
- Conceptual model makes it easier to create, edit, and alter models

The Map module provides tools for creating, managing and editing [[SMS:Feature Objects Types |feature objects]]. Feature objects are geometric entities, meaning they have a defined position or shape, and the attributes associated with those entities. The simplest feature object is a feature point, which defines a single location. Increasing in complexity, the next GIS object is an arc, which defines a line or polyline. Areas enclosed by polylines can be classified as a feature polygon.

Feature objects with related attributes are grouped into layers or coverages. The coverage definition includes a "type" that determines the attributes available for the objects in the coverage. See the list of coverage types to learn about the attributes associated with objects in that specific coverage.

The principal application of coverages in the SMS is to facilitate the representation of a numerical simulation in a representation that is independent of a specific discretization (a specific set of nodes and elements or cells). This allows the modeler to interact with a much smaller set of entities and reduces redundant effort in the modeling process.

A secondary application of coverages is to define geometric objects for data extraction from numerical model results.

Conceptual Model

A conceptual model is a high level representation used to define attributes used in the Mesh or Cartesian Grid generation process such as:

- Bathymetry source
- Materials information
- Boundary conditions

Creating Feature Objects

Feature objects are the building blocks of a conceptual model. They define the geometric shapes, locations and extents of objects in the model. There are several mechanisms for creating feature objects including:

- Extracting feature arcs from the contours of a scatter set. See Scatter Contour to Feature for more information.
- Importing from a web data source such as a coastline database. See Import from Web for more information.
- Importing from CAD data.
- Interactive definition (digitizing) using the Map Module Tools.
- Creation as a stamped feature to define built up embankment or dredged channels.

```
Normal   0         false   false   false
```

Elevations of Feature Objects

In the map module, Nodes, vertices and arcs all have an elevation attribute. That means that you can assign an elevation to individual nodes, points or vertices, or you can assign it to the arc. If you assign an elevation to an arc, the attribute of the nodes and vertices in that arc are updated as well. This will override any z-value specified for individual vertices or nodes on the arc.

The elevations of the map objects can also be assigned using the interpolation from the scatter module. In this case, each object (node, point, vertex, and arc) are assigned an elevation (z-value) based on the scatter set. The location for interpolation of the arc is the mid-point of the arc.

When you digitize in the map module, elevations are assigned as with other digitization in SMS. That means that when you create a node, point, or vertex, it is assigned the default elevation value for digitization. The default elevation is initialized to 0.0. The default changes any time you specify a Z-value. Therefore, if you create a map point or node, and specify an elevation for that selected point, the value you specify is now the default value for newly digitized points, nodes and vertices. (Note: when you create mesh nodes, there is an option to ask for an elevation each time you create a node, but this option is not available for scatter vertices or map module objects.)

When you convert a map coverage to a scattered data set, you have the option of using the arc elevations or the node and vertex elevations for the new scatter set. (There is also an option to use the arc spacing, but that is for a different purpose. It is not an elevation, but is useful sometimes as a function on a scattered data set.)

Map Module Overview

Map Module

Overview

The Map module provides tools for creating, managing and editing feature objects. Feature objects are geometric entities, meaning they have a defined position or shape, and the attributes associated with those entities. Feature objects with related attributes are grouped into layers or coverages. The coverage definition includes a "type" that determines the attributes available for the objects in the coverage. The principal application of coverages in the SMS is to facilitate the representation of a numerical simulation in a representation that is independent of a specific discretization (a specific set of nodes and elements or cells). This allows the modeler to interact with a much smaller set of entities and reduces redundant effort in the modeling process.

The map module also provides the tools to create and edit GIS like data and conceptual models as well as data for other purposes. Conceptual model is a geometry (mesh/grid) independent representation of the numeric model domain and/or boundary conditions and they can be converted to model geometry and boundary conditions. Conceptual model is also a high level representation used to define attributes used in the Mesh or Cartesian Grid generation process such as bathymetry source, materials information, and boundary conditions.

Functionalities

Map Module Display Options

Project Explorer

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a Map Module item.

Map Module Root Folder Right Click Menus

Right clicking on the Map module root folder in the project explorer invokes an options menu with the following options:

- New coverage – Opens the New Coverage Dialog.
- Clear Coverages – Deletes all coverages.
- Display Options – Opens the Display Options Dialog.

Coverage Item Right Click Menus

Right clicking on a Map item in the Project Explorer invokes an options menu with the following module specific options:

- Type – Change the coverage type.

Right click options for the coverage may also include options applicable only to the specific coverage type.

Menus

The following menus are available in the Map Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Feature Objects Menu

How do I?

To learn more about how to use the Map Module go to the Tutorials section.

Tools

The following tools are contained in the dynamic portion of the Tool Palette when the Map Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the map tool palette.

Tool	Tool Name	Description
	Select Feature Point or Node	The Select Feature Point or Node tool is used to select stand alone feature points or the ends of arcs. A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional points can be appended to the selection list by holding the SHIFT key while selecting by any method. Selecting new points without holding the SHIFT key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the SHIFT key as it is reselected. Pressing the ESC key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature points are locked so they are not accidentally dragged. When a single point is selected, its location is shown in the Edit Window. The coordinates can be changed by typing in the edit field. The Graphics Window's status bar will display information on the selected items depending on the settings find through the File | Info Options command in the File Menu. Selected points can be deleted by selecting the Edit | Delete menu command on the Edit Menu, by pressing the DELETE or BACKSPACE keys, or from the right-click menu. Arcs attached to the deleted points are deleted. This tool is available when one or more feature points exist.
	Create Feature Point	The Create Feature Point tool is used to place new feature point in the current coverage. A single point is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window. This tool is always available, however, creating a feature point is only allowed while in plan view.
	Select Feature Vertex	The Select Feature vertex tool is used to select one or more vertices on an arc. These vertices define the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Create Feature Vertex	The Create Feature vertex tool is used to create a new vertex on the interior of an arc. The vertex is created at the current arc location, but can be selected and moved to change the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Select Feature Arc	The Select Feature Arc tool is used to select one or more existing feature arcs. This is typically done to assign attributes to an arc or delete the arc. A single arc is selected by left-clicking directly on it. Double clicking on the arc will bring up the arc attributes dialog or that arc. Multiple arc can be selected at once by dragging a box. Additional arcs can be appended to the selection list by holding the SHIFT key while selecting by any method. Selecting new arcs without holding the SHIFT key will first clear the selection list and then add the newly selected points. A selected arc can be removed from the selection list by holding the SHIFT key as it is reselected. Pressing the ESC key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature arcs may have elevations associated with the arc as an entity. This is independent of the node and vertex elevations. When a single arc is selected, its elevation is shown in the Edit Window. The coordinates can be changed by typing in the edit field.

		Multiple feature arcs may also be selected to create a feature arc group to associate attributes with a string of arcs rather than a single arc. These arcs must connect end to end. The arc group is created from the Feature Objects menu command. The Graphics Window's status bar will display information on the selected items depending on the settings find through the File | Info Options command in the File Menu. Selected arcs can be deleted by selecting the Edit | Delete menu command on the Edit Menu, by pressing the DELETE or BACKSPACE keys, or from the right-click menu. Nodes attached only to the deleted arcs are deleted. This tool is available when one or more feature arcs exist.
	Create Feature Arc	The Create Feature Arc tool is used to create a new feature arc.
	Select Feature Arc Group	The Select Feature Arc Group tool is used to assign attributes to a string of arcs. The group must be created before it can be selected as a group. The attributes of the group then operate just as if the group was a single arc.
	Select Feature Polygon	The Select Feature Polygon tool is used to select polygons to assign attributes to them. The attributes associated with the polygon depends on the coverage type. Double clicking on the polygon will invoke the polygon attributes dialog for the polygon. Multiple polygons may be selected just as with arcs and nodes. The polygons must be created using the Feature Objects menu command.
	Select 1D Grid Frame	The Select 1D Grid Frame tool only appears when the coverage is associated with a 1D grid model (Cascade or GENESIS).
	Create 2D Grid Frame	The Create 2D Grid Frame tool is used to create a new grid frame for the creation of Cartesiona grids. The grid frame is defined by clicking three times in the graphics window. The first click defines the origin. The second click defines the I axis of the grid frame (both extents and direction). The third click defines the extents of the J axis. The direction is set to be perpendicular to the I axis.
	Select 2D Grid Frame	The Slect 2D Grid Frame tool allows the user to select a grid frame and alter its position, orientation and size.

Related Coverages

Generic Coverages:

- Area Property
- Mapping
- Observation
- Particle/Drogue
- Spatial Data
- Stamping
- Grid Frame

Model Specific Coverage

- ADCIRC
- ADH
- BOUSS2D
- CGWAVE
- CMS-FLOW
- CMS-WAVE
- Dredging
- FATE
- FESWMS
- GENESIS
- Generic 2D Mesh
- LTFate
- PTM
- STWAVE
- TABS
- TUFLOW

Grid Frame

For a Cartesian Grid coverage in the Map module, the *Grid Frame* menu item brings up the *Grid Frame* dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. The grid frame sets the bounds of the Cartesian grid that will be created when the user selects the *Map → 2D Grid* command.

Return to Grid Frame Dialog

3.7.a. Coverage Types

Coverages

Map Module

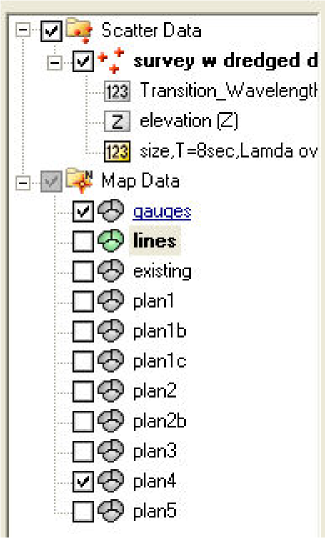
Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

Coverages

Feature objects in the Map module are grouped into coverages. Each coverage has a specific type, which determines the attributes that can be associated with geometric objects in that coverage. The coverages are grouped into conceptual models.

A coverage is similar to a layer in a CAD drawing. Each coverage represents a particular set of information. For example, one coverage could be used to define meshing zones and another coverage could be used to define zones of consistent roughness parameters. These objects could not be included in a single coverage since polygons within a coverage are not allowed to overlap and material zone boundaries don't necessarily coincide with meshing zone boundaries. Alternatively, one coverage could define Cartesian grid parameters for the same zone.

Coverages are managed using the Project Explorer. When SMS is first launched, a default coverage exists. If the user creates feature objects they are placed in the current coverage. When multiple coverages are created, one coverage is designated the "active" coverage. New feature objects are always added to the active coverage and only objects in the active coverage can be edited. The figure below shows several coverages in the Project Explorer. The active coverage is displayed with a green colored icon and bold text. A coverage is made the active coverage by selecting it from the Project Explorer. In some cases it is useful to hide some or all of the coverages. The visibility of a coverage is controlled using the check box next to the coverage in the Project Explorer. In the example below, several design options are not displayed.



Creating a New Coverage

A new coverage can be created by right-clicking on the Map Data folder in the Project Explorer. Select New Coverage from the pop-up menu. The New Coverage dialog will appear and the coverage type will need to be defined. Each coverage type is organized according to whether it is a Generic or a Model type coverage. If a coverage type was selected that has attributes associated with it, they can be changed by clicking the Attributes button. If there are no attributes associated with the selected coverage type the Attributes button will be unavailable. The name of the coverage is also specified here. After clicking the OK button, the new coverage will appear in the Project Explorer.

Coverage Right-click Menu

Right-clicking on a coverage brings up a menu with the following options:

- **Delete**
- **Duplicate**
- **Rename**
- **Convert** – Coverage feature object data can be mapped to other geometric objects or numerical models by selecting one of the Map -> ... options in the Convert submenu.
- **Coordinate Conversion** – The Coordinate Conversion dialog opens, which allows the user to specify a coordinate conversion to be performed on the coverage data.
- **Metadata** – Metadata associated with this coverage can be edited and viewed.
- **Zoom to Coverage** – Frames the graphics window to the extents of the data displayed in the selected coverage.
- **Type** – Sets the coverage type.

Merging Existing Coverages

Occasionally, a user may want independent features of two separate coverages combined into one converage. SMS allows the user to merge these two coverages together. The user should left click one of the coverages listed in the data tree then multi-select the other coverages to be merged. This can be done one at a time by holding the CTRL key, or several adjacent coverages can be selected by holding the SHIFT key and then clicking the last adjacent dataset.

Once all datasets to be merged have been selected, the user simply accesses the right-click menu and chooses "Merge Coverages". A dialog may appear asking the user if they wish to delete the coverages used to create the merged coverage.

Coverage Types

The attributes of entities on a coverages belong to a list of attributes associated with a type of coverage. For example, arcs in mesh coverages have boundary conditions compatible with the specific finite element model they are associated with, and polygons in those coverages include attributes associated with meshing and material types. The following coverage types include the following:

- Generic Input (Preprocessing)
  - Area Property
  - Feature Stamping
  - Spatial Data
- Generic Output (Postprocessing)
  - Observation
  - Particle/Drogue
  - Shoals
- Generic Model Interface
  - Mapping
- Models
  - ADCIRC
  - ADH
  - BOUSS-2D
  - CGWAVE
  - CMS-FLOW
  - CMS-WAVE
  - Dredging
  - FATE
  - FESWMS
  - GenCade
  - GENESIS (replaced by GenCade)
  - Generic Mesh
  - LTFate
  - PTM
  - STWAVE
  - TABS
  - TUFLOW

Related Topics

- Calibration Targets
- Map Module

3.7.a.1. Generic Coverages

Generic Coverages

Generic Coverages:

- Area Property
- Mapping
- Observation
- Particle/Drogue
- Spatial Data
- Stamping
- Grid Frame

Related Topics

- Map Module

Area Property Coverage

An area property coverage is used to map properties such as Manning's roughness values to the mesh, grid, or cross-sections.

Materials are used to define different values for the property the material represents. For example, if the material represents the Manning's roughness value, materials are defined to represent the different Manning's roughness values to be included in the model (stream, floodplain, field, roadway, etc.). Polygons can then be created to define the stream, floodplain, field, and roadway with the corresponding material assigned.

Materials are assigned a name, color and pattern for display, and model specific attributes.

Many of the data entities constructed and edited in SMS (i.e., elements, cells) have a material ID associated with them. This material ID is an index into a list of material types. Materials contain model specific parameters such as manning's roughness, or bed material grain size. A global list of material attributes is maintained and can be edited using the menu command *Edit* | *Materials*. This command brings up the *Materials Data* dialog where each material is assigned an ID number. This dialog can be used to delete unused materials, create new materials, and assign a descriptive name, color, and pattern to a material. This general information is saved in the material file. The materials defined within the Materials Data dialog are available for all modules. Contents

Dialog Description

The *Materials Data* dialog is accessible from the menu command *Edit* | *Materials* or from model specific material properties windows (ex. ADH), available in the model specific menu. The dialog is resizable by dragging on the window edges.

When a new mesh element or grid cell is created, the material is assigned to the new object based on the materials options in the *Element Options* dialog.

Model specific material properties such as Manning's n and Eddy viscosity are edited using commands available in the model specific menu.

Materials Spreadsheet

The materials spreadsheet contains three columns (ID, Name, and Pattern) for the defined materials. All IDs must be unique and the spreadsheet can be sorted by clicking on the column headings. The default *Disable* material cannot be edited (except the display pattern) and will always be at the top of the spreadsheet regardless of sorting. Each material is accompanied by a pattern button in the *Pattern* column. To select a pattern, click on the preview section (left side) of the button to open the *Pattern Attributes* window. To quickly edit only the color, click on the down arrow (right side) of the button, and make a selection in the pop up color palette.

Buttons

- New – Inserts a material into the spreadsheet with the lowest unique ID available and a default name and pattern.
- Delete – Removes the currently selected material from the spreadsheet.
- Copy – Creates and inserts a copy of the currently selected material with the lowest unique ID available and a default "copy of" name.

Legend

- Legend – Check box with the associated Options... button controls the display of a legend of the materials in the *Graphics Window*.
- Options... – Opens the *Legend Options* dialog. The options for the legend are edited in the *Legend Options* dialog. These options include:
- The name to be displayed on the legend.
- The specification of where on the screen the legend will appear.
- The font to be used in the legend.
- The size of each entry in the legend.

Note: only active materials are included in the legend.

Area Property Polygon Attributes Dialog

The *Land Polygon Attributes* dialog is used to set the attributes for feature polygons. Attributes that can be specified for each polygon include:

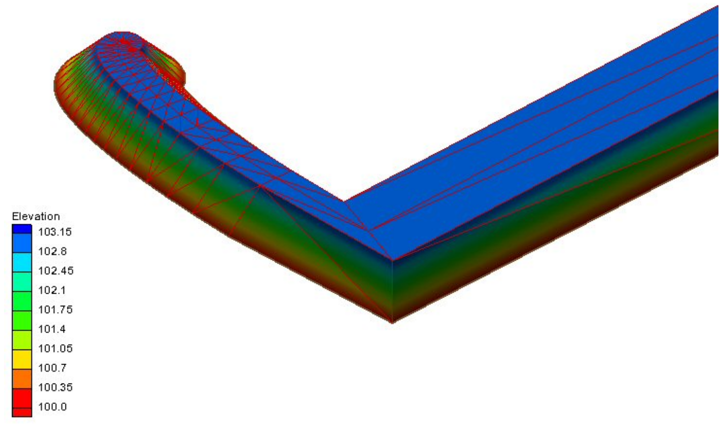
- Polygon Type
  - None
  - Material

Materials must be defined in the *Materials Data* dialog before they can be assigned to a polygon.

Related Topics

- Coverages

Feature Stamping



A stamped embankment with a guidebank

Feature stamping is the terminology used to refer to the insertion of man made structures into a natural topography or bathymetry set. In common terms, this means adding an embankment (such as a levy) or dredging a channel. A stamped feature usually follows a linear object or centerline. However, it can also be based around a single point to create a mound or pit, or applied to only one side of a closed line (a polygon) to create a flat topped mound or flat bottomed pit.

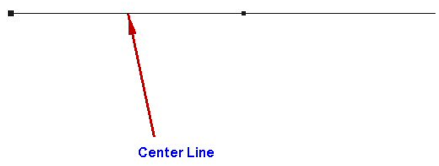
The Process

The basic steps to define a linear stamped feature include:

1. Define the stamping coverage and centerline (or focal point) of the stamped feature
2. Assign attributes to the centerline including
    1. The elevation along the centerline
    2. The cross sections along the centerline
3. Stamp the feature. This converts the centerline and its attributes to:
    1. Another coverage containing all the extents and details of the feature
    2. A scatter set defining the elevation for the feature.

Sample problems in the section below illustrate the procedure.

Define the Coverage and Centerline



A centerline arc – simple

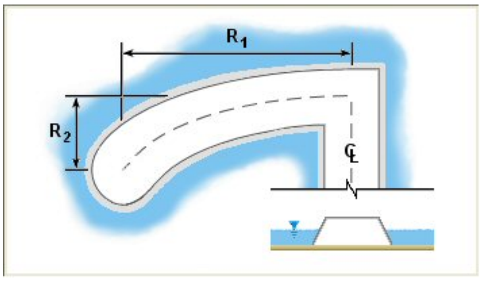
SMS utilizes a coverage of type "Stamping" to create the stamped features. Depending on the application, you may want to have multiple "Stamping" coverages to represent different design alternatives. Each coverage may contain multiple features. Create a "Stamping" coverage by right clicking on the "Map Data" entry in the **Project Explorer** and selecting the **New Coverage** command. Then right click on the new coverage and set its type to "Stamping".

The ambient geometry is defined by a scatter set (and one of its associated datasets). This surface determines the cut-off for the sloped banks of a stamped feature. Right clicking on the coverage and selecting "Properties" brings up the dialog that associates a specific data set to the stamping coverage. You must also specify whether this surface is defined as elevations (positive up) or depths (positive down). By default, SMS interprets this surface as elevation data.

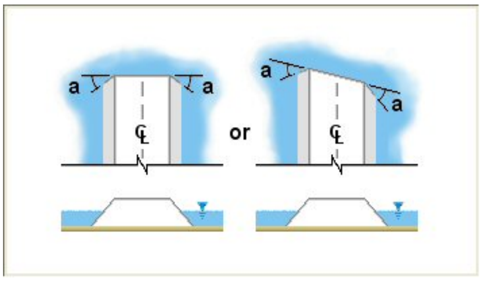
Define Feature Attributes

Any arc or point created in a "Stamping" coverage has attributes to create a stamped feature. Attributes are assigned in the Stamping Point Attributes Dialog or Stamping Arc Attributes Dialog. Attributes include:

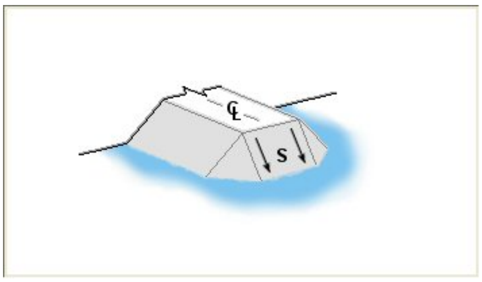
- A feature name. This will be used when SMS creates stamped feature coverages and scatter sets from the stamping coverage.
- The feature type (cut = channel or fill = embankment). If you have a stamped feature that contains both cut and fill sections, create one coverage as a cut, then duplicate the coverage and change the copies type to fill.
- A base elevation (the top elevation of the embankment or the bottom of the channel). This can be specified from the node/vertex elevations, as a constant, or extracted from a curve of elevation -vs- distance.
- The cross sectional shape(s). The cross section can be defined as a template, which is propagated all along an arc, or can be individually specified at each vertex in an arc. The cross section can also be defined individually for the left and right side of the arc. One point on each side of the cross section can be specified as the "shoulder". For a channel, this would be the "toe", but the reference in SMS is the shoulder point. When the arcs representing the shoulder are created, the user has the option to create an arc along this shoulder. If vertical walled structures are desired, the cross section can simply stop at the edge (shoulder). This will result in a feature arc at the edge and a scatter set for the top of the structure.
- The method for treating the ends of the structure. The options include:
  - Wingwalls
  - A sloped abutment (spillthrough)
  - A guidebank



Guidebank



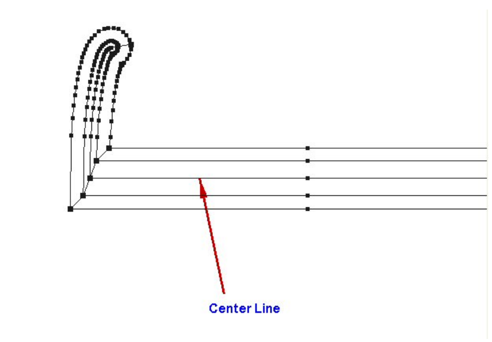
Wing Walls



Spillthrough

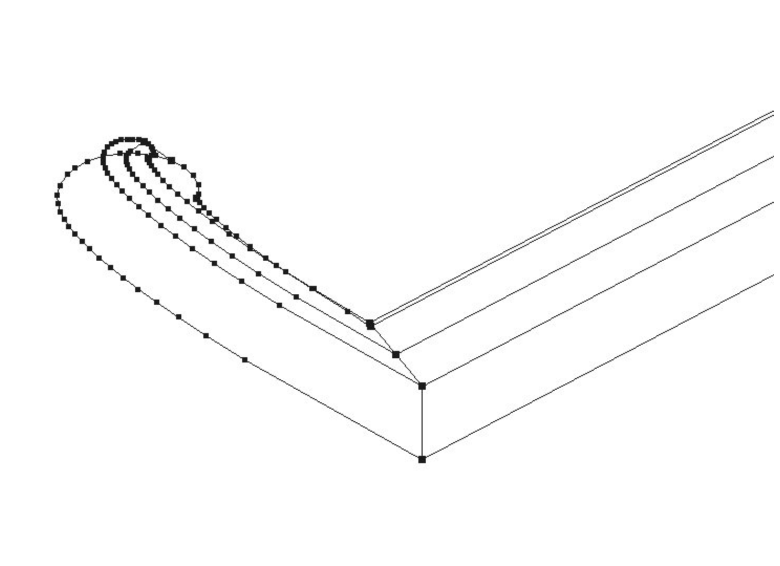
Stamping the Feature

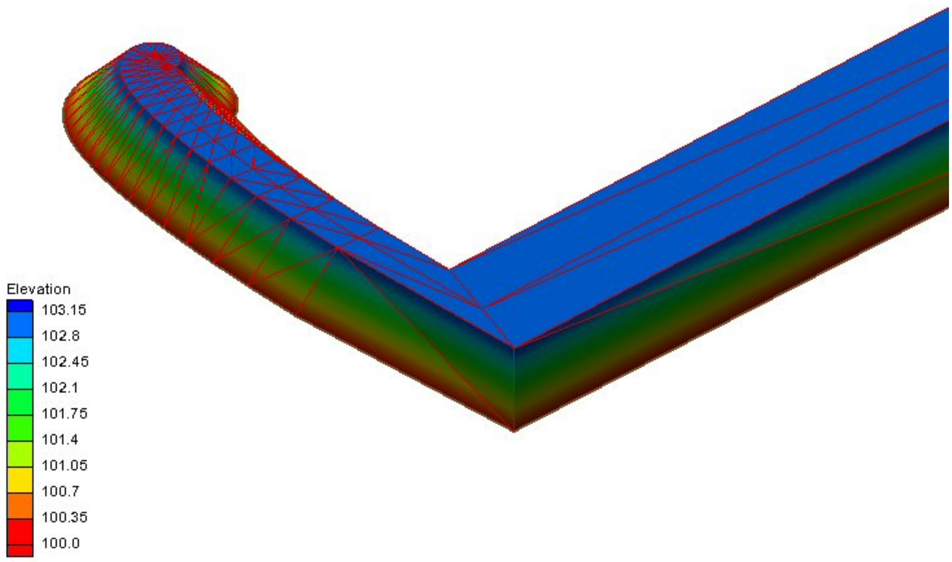
To create the stamped feature, right click on the "Stamping" coverage and select "Convert → Stamp Features ...". This brings up a dialog that allows the user to specify what output should be created from the process. Specifically, the process can create:



Stamped Feature Arcs – Created from Centerline

- A new scatter set which defines the elevation points for the new structure. This is named based on the stamped feature name. SMS triangulates all these points to create a surface and trims the scatter set to the extents of the stamped feature. The arcs that make up the stamped feature are converted to scatter breaklines to assist in the triangulation.
- A new coverage which defines the breaks and extents of the new coverage. The dialog allows the user to specify the type of the coverage. Generally, it is recommended that the coverage type be selected to match the numeric engine that will evenutally be used for the simulation. Options also exist to determine whether the stamped feature will include the center line, the shoulders, and the cross sections. The extents of the stamped feature are always generated.


Stamped Feature Arcs – Oblique view



Stamped Feature Scatter Set – Oblique view

Case Studies / Sample Problems

There are a wide variety of stamped features that can be created using this tool. The Feature Stamping tutorial in the general section of the tutorials may be helpful for learning to use them.

- Embankments on a flat plain
  - Vertical sides on an embankment
  - Sloped sides on an embankment

External Links

- Emery, R. N. (2005). Refining and Expanding the Feature Stamping Process. Thesis, Brigham Young University. [1]
- Christensen, J. R. (2001). Stamped Features: Automatic Generation of Flow Modifying Structures in Conceptual Models. Thesis, Brigham Young University.
- Zundel, Alan K. and J. Ryan Christensen, "Stamped Features: Creation of Engineered Structures in Conceptual Models", International Journal of Hyrdroinformatics, Vol. 4, No. 1, February 2002, pp. 63-72.

References

[1] http://contentdm.lib.byu.edu/ETD/image/etd1004.pdf

Observations

SMS contains what is called an observation coverage that is designed to help in model verification and calibration processes. Result verification is an important part of the computer modeling process. SMS includes a number of powerful tools, associated with an observation coverage, that allow users to verify simulation results with observed data. The two tools used for verification and calibration in an observation coverage are observation points and observation arcs. Observation points are used to verify the numerical analysis with measured field data such as water surface elevation or velocity data. They are also be used to see how computed values change with time at a particular location. Observation arcs are used to view the results at a cross section or along the river profile. These tools can be used with any of the SMS models.

Creating an Observation Coverage

To create a new observation coverage:

1. Right-click the *Map Data* item in the *Project Explorer*
2. Select *New Coverage* from the right click menu
3. Set the coverage type to *Generic → Observation* in the *New Coverage Dialog*
4. Set the coverage name as desired
5. Click *OK* to exit the dialog

Alternatively, an existing coverage can be changed to an observation coverage by right clicking on the coverage in the Project Explorer and setting the type to *Observation* using the right click menu.

Creating an Observation Point

Observation points are created at locations in the model where calibration data such as the velocity or water surface elevation has been measured in the field. Each observation point is used to compare the measured values with the values computed by the model at the point's x, y location. This comparison can assist the modeler in determining the accuracy of the numerical model results. If the numerical model results do not match the observed field data, model parameters such as manning's roughness may need to be modified to obtain more accurate results.

Creating an observation point is just like creating a feature point in any other coverage type. Select the Create Feature Point tool from the Dynamic Toolbar and click the location for the feature point.

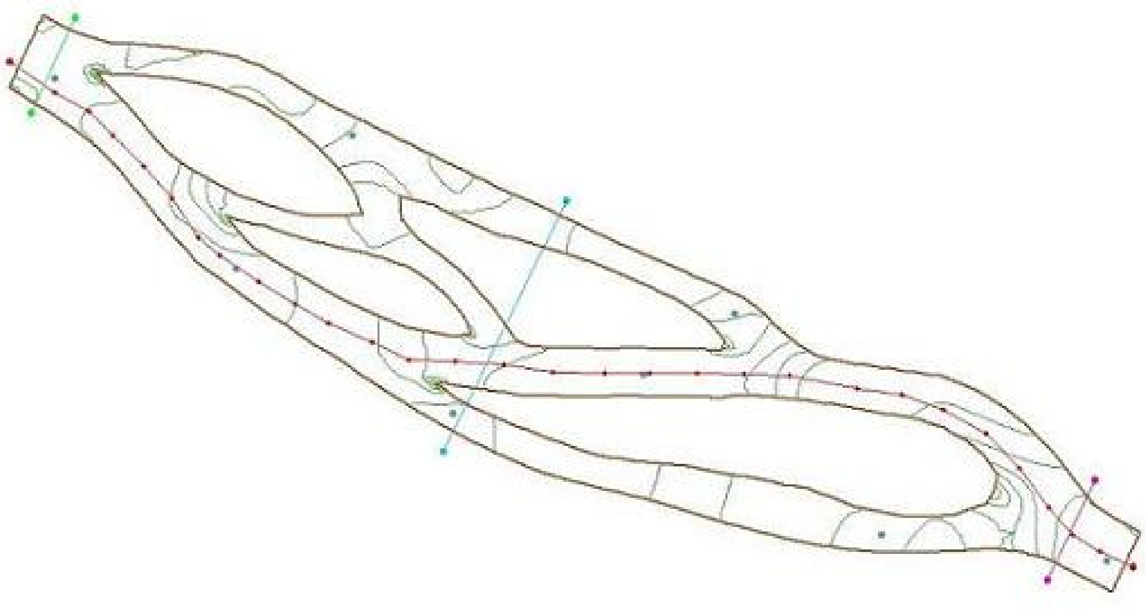
Creating an Observation Arc

Observation arcs are created at cross sections in the model where calibration data such as the flowrate has been measured in the field. Observation arcs compute fluxes across the arc. Therefore, measurements for observation arcs are called *Flux Measurements*. Each obseration arc is used to compare the measured values with the values computed by the model across the vertical plane defined by the arc. This comparison can assist the modeler in determining the accuracy of the numerical model results. If the numerical model results do not match the observed field data, model parameters such as manning's roughness may need to be modified to obtain more accurate results.

Creating an observation arc is just like creating a feature arc in any other coverage type. Select the Create Feature Arc tool from the Dynamic Toolbar and click out your arc. Double-click to end the arc. In an observation coverage, profile arcs and cross section arcs may be useful to analyze a simulation's solution.

Setting Observation Object Attributes

Observation point and arc attributes are defined in the Observation Coverage dialog. See Observation Coverage dialog for a description of the Observation Coverage dialog.



Viewing Results

In addition to viewing the results of the solution data versus the observed data on the calibration targets, additional plots can be created using the Plot Wizard. See Plot Window for a description of the available plot types.

Calibration

Calibration is the process of modifying the input parameters to a model until the output from the model matches an observed set of data. SMS includes a suite of tools to assist in the process of calibrating a model. Both point and flux observations are supported. Most of the calibration tools can be used with any of the models in SMS.

Measurement types can be defined in SMS. They are defined in the map module and are associated with points and arcs in an observation coverage. Point observations represent locations in the field where some value has been observed. In most cases, the points will correspond to gauges or high water marks and the value will be the elevation of the water (the head) or a flow velocity (and possibly direction). Flux observations represent linear or areal objects such as streams gages where the flow rate has been measured or estimated. Both point and flow observations can be assigned a confidence interval or calibration target.

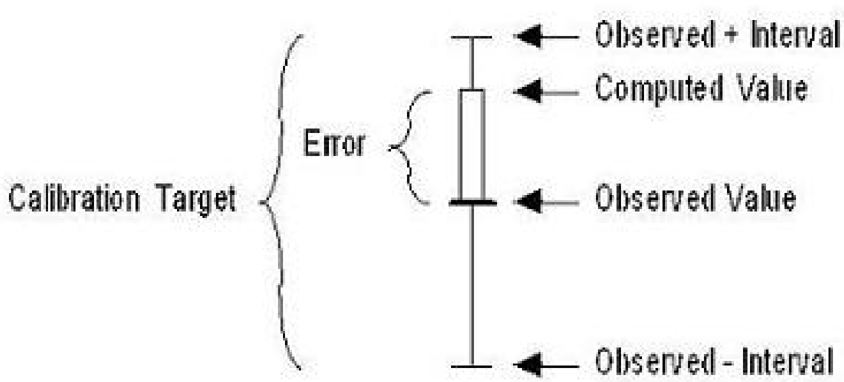
Once a set of observed point and flow values has been entered, each time a model solution is imported, SMS automatically interpolates the computed solution to the observation points. A calibration target representing the magnitude of the residual error is displayed next to each observation point and each flux object as shown below. The size of the target is based on the confidence interval or the standard deviation. In addition to the calibration targets next to the observation points, you can choose to display any of a number of statistical plots.

Calibration Target

If an observed value has been assigned to an observation point or if an observed flow has been assigned to an arc or polygon, the calibration error at each object can be plotted using a "calibration target". A set of calibration targets provides useful feedback on the magnitude, direction (high, low), and spatial distribution of the calibration error.

The components of a calibration target are illustrated in the following figure. The center of the target corresponds to the observed value. The top of the target corresponds to the observed value plus the interval and the bottom corresponds to the observed value minus the interval. The colored bar represents the error. If the bar lies entirely within the target, the color bar is drawn in green. If the bar is outside the target, but the error is less than 200%, the bar is drawn in yellow. If the error is greater than 200%, the bar is drawn in red. The display options related to calibration targets are specified in the Calibration Display Options dialog.

If the active time step is before the first observed time, or after the last observed time, the targets are drawn lighter.



Calibration Display Options

Calibration targets are drawn next to their corresponding map data (point, arc, polygon). The Feature Objects Display Options dialog contains a toggle labeled calibration targets. Below the toggle is a Scale edit field.

The target is drawn such that the height of the target is equal to twice the confidence interval (+ interval on top, - interval on bottom). The Scale edit field allows the user to change the general length and width of the targets independent of the range of the active dataset.

Observation Coverage Dialog

The *Observation Objects* dialog has two sections used to define the attributes of the points and arcs created in the observation coverage. To open the *Observation Coverage* dialog:

1. Make the observation coverage active in the Project Explorer
2. Select the Create Feature Point or Create Feature Arc tool from the Dynamic Toolbar
3. Select a feature point or feature arc
4. Select the menu command *Feature Objects* | *Attributes* or double click in the previous step

Dialog Layout

The options in the dialog will differ slightly based on the feature object type currently being edited (arc or point). The feature object type is specified using the combo box in the upper right of the dialog.

In addition to a unique Name and Dataset(s), two other parameters are used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the *Trans* toggle. The Module of a measurement refers to the SMS module where the computed data is stored. (The Module is set by default and normally does not need to be changed.)

Observation Point Attributes

When "points" is selected in the Feature object type combo box, the top section of the dialog is entitled "Measurements" and the bottom section is entitled "Observation Points." The *Measurements* section is used to specify which data sets correspond with the observed data entered in the *Observation Points* section. The *Measurements* section is used to enter the observed data at each point. Each observation point is assigned the following attributes:

- **Color –** Color of the observation point.
- **Observe –** Turn the observation point on or off.
- **Name –** Name of the observation point.
- **X –** X-location of the point.
- **Y –** Y-location of the point.
- **Observed Value –** Value measured in the field corresponding to the active measurement.
- **Interval –** Allowable error (±) between the computed value and the observed value. Model verification is achieved when the error is within the interval (±) of the observed value.
- **Angle –** When a measurement for observation points is tied to a vector data set (as is the case with a Velocity measurement) an angle needs to be specified. This angle is an azimuth angle with the top of the screen representing north when in plan view.

Observation Arc Attributes

If "arcs" is selected in the combo box, then the top spreadsheet had the name "Flux Measurements" and the bottom spreadsheet has the name of "Observation Arcs."

To define an arc measurement, the Name must first be defined. In addition to a unique Name, a scalar and vector data set must be assigned to it. Two other parameters are also used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the Trans toggle. The Module of a measurement refers to the SMS module where the computed data is stored. (The Module is set by default and normally does not need to be changed.)

Each observation arc is assigned the following attributes:

- **Color –** Color of the observation arc.
- **Observe –** Turn the observation arc on or off.
- **Name –** Name of the observation arc.
- **x-origin –** X origin of the arc (used to modify the x-value used in plots).
- **Observed Value –** Value measured in the field corresponding to the active measurement.
- **Interval –** Allowable error (±) between the computed value and the observed value. Model verification is achieved when the error is within the interval (±) of the observed value.

Related Topics

- Coverages
- Plot Window

Particle/Drogue

A particle/drogue coverage is used for Visualization|post-processing. The feature points and arc nodes/vertices define the seed locations for generating an animation of particles flowing through a hydrodynamic current. The particles are simulated as massless objects or "drogues" floating in the flow field. SMS computes the paths these particles would follow when driven by the currents of the flow field using numerical integration.

This coverage must be defined before selecting the drogue option when setting up a film loop.

When displaying the resulting animation, the particles may be displayed in a color based on the current velocity of the particle or the distance the particle has traveled. The user also specifies the length of the tail behind the particle (in units of time). Therefore, a longer particle tail indicates a generally faster moving particle.

Drogue plot animations are different from flowtrace animations in that the distances traveled by each drogue represents the actual physical speed of the flow field.

Application

- Residence time calculations. One of the most useful applications of drogue plots is to approximate residence time of a basin or other subregion of a hydrodynamic domain. In order to approximate this, the user would:
  - Distribute a fairly large number of drogue seed points inside the basin (or sub domain) of interest).
  - Generate a drogue plot film loop of the time range of interest. (Currently the hydrodynamic solution must include enough time steps to span the residence time in question. If this is not the case, additional time steps should be added to the data set either by rerunning the simulation for a longer time period or duplicating time steps).
  - Review the number of particles still in the basin (or subdomain) at various time intervals. The percentage of particles, and their position give an indication of overall concentration and local concentration in the basin.
- General visualization of flow field.

External Links:

- “Gas Flow Visualization for Combustion Analysis”, Energy and Fuels, Vol. 7, No. 6, 1993, pp. 891-896. Zundel, A.K., Saito, T., Owen, S.J., Sederberg, T.W., Christiansen, H.N. [1]

Related Topics

- Coverages
- SMS:Animations

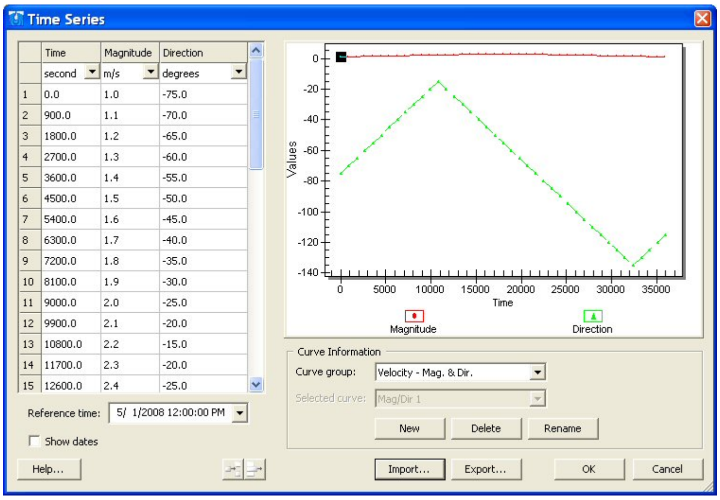
References

[1] http://pubs.acs.org/doi/abs/10.1021/ef00042a027

Spatial Data

The Spatial Data coverage is designed to store, visualize, and analyze various types of data at node locations. Most commonly, this data would consist of time series curves. The data can be accessed or added to a node by the right-click menu of a single selected node. From this menu the data associated with the node can be created, viewed, modified, or deleted.

The following types of data are supported (right click options are described):



Time Series Editor

Time Series – The time series editor allows the user to view, edit and import/export transient datasets for the location. Each time series consists of either a scalar value at each time, or a vector value at each time. The vector quantities may be defined as (X,Y)components or ((Magnitude,Direction) pairs. The time values may be displayed as dates or offset values. The time series are stored in a database inside SMS. When the project is saved, the series are saved in the XMDF project file. The series may also be imported/exported using a TSD file format..

- *Edit Data* – This menu option invokes the time series editor. The list of time series curves available for this node are listed in a list box. The user selects the desired time series curve, and the values for that curve appear in the spread sheet. The selected time series data may be modified in this spread sheet.
- *View Data* – This menu option operates like the Edit Data option, but the spread sheet is set to read only. This prevents accidental modification of the values in the time series.
- *Delete* – This option removes the association between the selected time series and the spatial data node. The time series curve remains accessible in the Time Series database.

- **Compass Plot**
  - *Show Connection Lines* – This option allows the lines connecting the compass plot to its associated spatial data node to be enabled or disabled.
  - *Properties...* – This option brings up the Compass Plot Properties dialog for the selected compass plot.
  - *Delete* – This option deletes the selected compass plot.

The Spatial Data coverage type is found in the "Generic" coverage type list.

Compass Plot

Compass Plots may be created on a *Spatial Data* node. The *Compass Plot* displays arrows to represent temporally varying vector data. This plot can be used to show a vector quantity, either varying through time or constant, to illustrate quantities such as wind direction, wave direction, or current direction. The plot is associated with a specific point, but that point does not have to be part of a numerical model or physical object. It could be created simply to hold the prevailing wind direction, for which a compass plot would be created.

Direction Convention

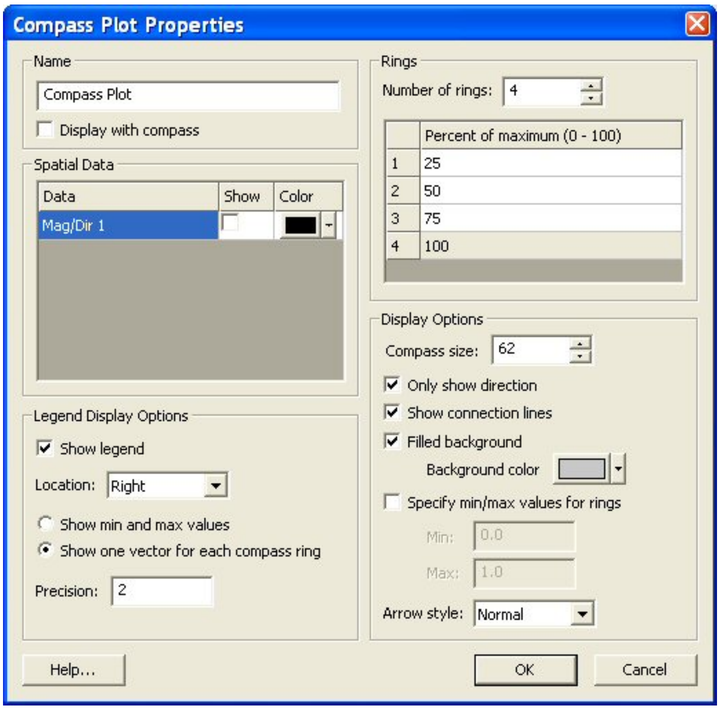
For curves with a specified direction, the compass plot uses a "TO" direction convention with North being 0.0 and the angle is measured clockwise. This means that a vector with a direction of 0.0 points North, 90 degrees points East, 180 degrees points South and 270 degrees points West.

Layout

When a plot is created, SMS places on the left side of the screen. The user may select the plot using the "Select Compass Plot" tool  and position it at any location. It is positioned in screen space, so as the user pans or zooms around the modeling domain, the plot stays in a single location. The plot can also be resized graphically or using its attributes dialog.

Creating a Compass Plot

- In order to create an use a compass plot, a user will perform the following steps:
  - Create or Select a *Spatial Data* Coverage.
  - Create or Select a *Feature Point* in the *Spatial Data Coverage*.
  - Make sure at least one vector time series curve is stored for the selected feature point.
  - Right click on the point and select the Add → Compass Plot command. This creates a compass plot and brings ups the *Compass Plot Properties* dialog. Each time series to be included in the compass plot must be selected in the "Spatial Data" section of the dialog. Clicking *OK* will cause the dialog to disappear and the compass plot to appear. Properties of the compass plot include:
    - The name of the plot which can optionally be displayed at the top of the compass plot.
    - A flag to show/not show a vector for each vector time series stored at the *Spatial Data* point.
    - Options for a compass plot legend, including:
      - A flag to show/not show the legend.
      - Set the position of the legend. Options include any side of the plot.
      - The number of vectors to show in the legend. This can be the min/max, or one for each compass ring.
      - The number of digits of precision for the legend.
    - Control of the number of rings to show in the plot, and the percent of maximum value for each ring. By default SMS creates the compass plot with four concentric circles, representing 1/4, 1/2, 3/4 and the maximum vector magnitude.
    - Display options including:
      - The pixel size of the compass plot.
      - A flag to show only the vector direction (ignore magnitude).



Compass Plot Properties

- A flag to show connection lines. Since the plot can be selected and drug to any position on the screen, these lines can be useful to show a location the vectors apply to.
- A flag to set the background color (if desired) for the compass plot. By default, the plot is filled with a gray background.
- An option to specify the magnitude range to display. Any vector with magnitude above the maximum will appear as a 100% magnitude vector. Below the minimum, the vector will not be displayed.
- The vector style. (This is a future enhancement. Currently, only "Normal" is supported.

Editing a Compass Plot

- In order to edit or adjust a compass plot, a user will perform the following steps:
  - Select the *Select Compass Plot* tool  .
  - Left-click on the selection box of a compass plot and drag to position and resize the plot.
  - Right click on the plot itself. I menu appears including the following options:
    - Delete – This allows the compass plot to be deleted
    - Show Connection Lines – this sets the display option of the plot
    - Show Legend – this turns on/off the legend for the compass plot
    - Legend Location – this is a pull right menu that allows the user to position the legend
    - Properties – this invokes the compass plot properties dialog to allow the user to edit any of the attributes.

Related Topics

- Time Series
- Spatial Data Coverage
- Coverages

Shoals

SMS includes the ability to import and export tabular data files. These files can include any number of columns of data. The user selects which columns are to be imported and how they are to be interpreted. The columns of data follow an optional header and may be delimited by any character (such as TAB, SPACE, COMMA, etc.). Files with the extensions of *.xyz and *.pts are defaulted to be of this type.

History

This capability was originally developed to support files generated by the SHOALS group of the US Army Corps of Engineers [1]. SHOALS files generally have an optional file header describing the data in each column of the file.

Importing Tabular Data

Tabular data files are opened using the File | Open menu command. The user selects the tabular data file to open. The Text Import Wizard appears and allows the user to select how the data should be interpreted (as scatter points, mesh nodes, or map nodes) and which columns should be imported.

Exporting Tabular Data

Tabular data files are saved using the File | Save As... menu command and selecting *Tabular Data Files (*.txt)* or *Shoal Files (*.pts)* from the save as type filter combo box.

When in the Scatter module this saves scattered data vertices to the tabular file. When in the Mesh module this save the mesh nodes into the tabular data file. In either case the Export Tabular File dialog appears to support this operation.

Related Topics

- File Formats

References

[1] http://www.erdc.usace.army.mil/

3.7.a.2. Model Specific Coverages

Model Specific Coverages

Model specific coverages

- 1D Hyd Cross-Section Coverage
- 1D River Conceptual Model
- ADCIRC
- ADH
- CGWAVE
- CMS-Flow
- CMS-Wave
- Dredging
- FATE
- FESWMS
- GenCade
- Generic Mesh Coverage
- LTFate
- PTM
- STWAVE
- ADCIRC Wind Coverage
- BOUSS2D
- EFDC
- TABS (RMA2/RMA4)
- TUFLOW Coverages

Related Topics

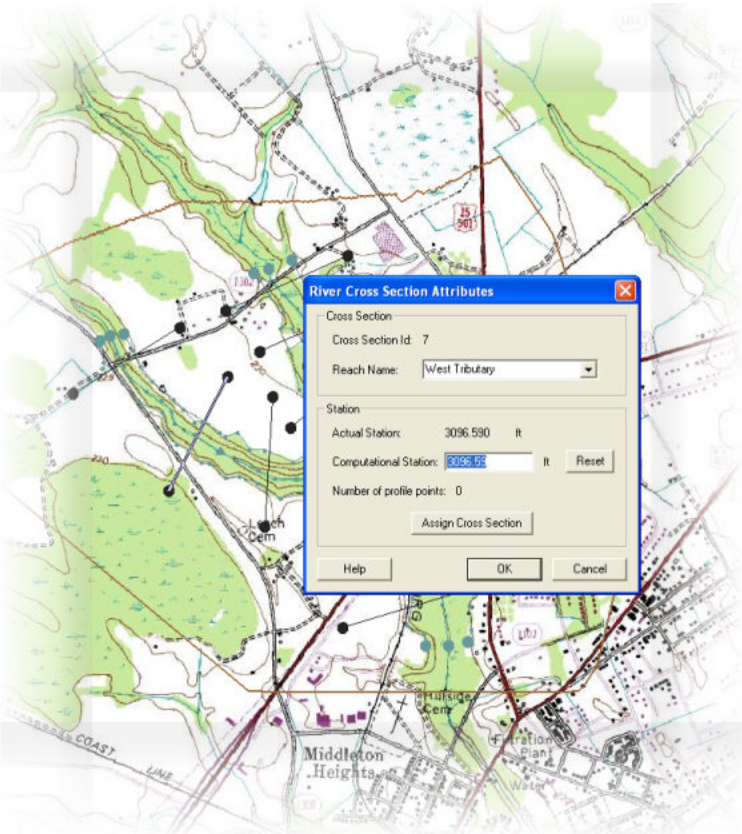
Calibration Targets Map Module

1D Hyd Cross-Section Coverage

The 1D-Hydraulic Cross-section coverage is used to identify the cross section stations in the hydraulic model, and can also be used to automatically cut a cross section from an underlying digital terrain model. The attributes of a cross-section feature arc is the cross-section itself, along with the other parameters that define its topology in the model and include: a cross-section ID (internally assigned), the reach name (inherited from the centerline arc it intersects), the station (inherited from the centerline), and any specific model attributes. The 1D-Hydraulic coverage is used in conjunction with the cross sections and digital terrain model in order to determine the thalweg position (from the centerline arc) and the left and right bank points (from the bank arcs).

A cross-section is assigned automatically when cutting the cross sections, or can be assigned manually (imported from a file or entered directly) using the cross section editor.

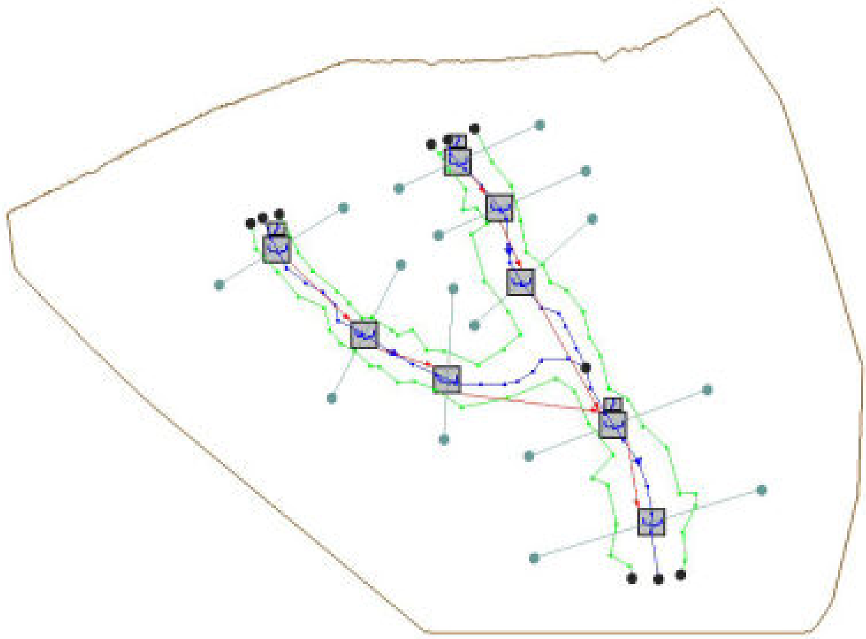
See the help for Editing Cross Sections to learn more about how cross sections are managed and edited.



Return to 1D River Conceptual Model

1D River Conceptual Model

SMS uses a conceptual model (coverages of centerlines, and cross-sections) to define the hydraulic model, but at some point this conceptual model must be mapped to an equivalent topologic model representation for a Hydraulic model. HEC-RAS, for example, is defined as reaches and cross sections. Each reach and section has appropriate stationing defined from the spatial nature of the conceptual model. When choosing this command a schematic of the river is made for the appropriate model. The example below shows the reach (small boxes at the beginning of each reach) and section icons of the schematic for an HEC-RAS model. In the Hydraulic Modeling module the schematic is generally used as the view of the model.



Related Topics

- 1D Hyd Cross-Section Coverage
- 1D River Module

ADCIRC

An ADCIRC coverage is used to build a conceptual model of an ADCIRC project. The conceptual model defines parameters such as model extents, mesh generation options, and boundary conditions.

ADCIRC Conceptual Model Development

The following steps are generally followed when creating an ADCIRC conceptual model:

Define Coastline

The coastline can be defined in any of the following manners:

- Read in an existing coastline file (*.cst) (see Create Coastline). Coastline files include lists of two-dimensional polylines that may be closed or open. Open polylines are converted to Feature Arcs and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.
- Extract a coastline arc from a scatter set using the Scatter Contour to Feature command.
- Digitize a coastline arc using the Create Feature Arc tool.

Define Ocean Boundary

Once the coastline has been created, the ocean boundary can be defined in any of the following manners:

- Use the Define Domain menu command to automatically generate the ocean boundary.
- Digitize the ocean boundary arc using the Create Feature Arc tool.

The ocean boundary can take on a rectangular, semi-circular, or circular shape depending on your coastline form. This will close the domain for your project, giving you a defined area where a finite mesh can be created and the ADCIRC model can perform its analysis.

Build Polygons

Build polygons using the Build Polygons menu command.

Choose Mesh Generation Method

At this point, a choice must be made to generate the mesh using the LTEA Toolbox or manual mesh generation methods.

LTEA Toolbox

The LTEA Toolbox can also be used to generate a mesh from a bathymetry scatter set and the ADCIRC coverage created in the previous steps.

Manual Mesh Generation

To manually generate a mesh:

- Use the Select Feature Polygon tool and double click on a polygon to open the 2D Mesh Polygon Properties dialog. You can also use select a polygon and then select "Attributes" from the right click menu or select a polygon and use the Attributes menu command to open the 2D Mesh Polygon Properties Dialog.
- Set the desired mesh options. See the article on mesh generation for an explanation of mesh generation.

Related Topics

- Define Domain
- Coverages
- ADCIRC
- Boundary Conditions
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes
- Steering

ADCIRC Wind Coverage

The ADCIRC wind coverage represents a storm, such as a tropical depression or hurricane. This information can be imported from NWS 8 and 9 files (fort.22 and fort.15 files), and exported by SMS for use with ADCIRC. Running ADCIRC directly from SMS also works fine, and various model checks have been implemented to help users catch mistakes and avoid crossing ADCIRC's limitations. To create a wind coverage, create a new coverage and select the coverage type Models | ADCIRC | Wind. Alternately, you can convert an existing coverage by right-clicking it and selecting Type | Models | ADCIRC | Wind.

Coverage Attributes

To access the coverage attributes, right-click on the coverage and select "Properties...". The coverage attributes dialog contains several separate fields that apply to the entire hurricane, such as whether it is symmetric or asymmetric. The storm's symmetry in particular is important to set before editing node properties because it affects which fields are shown by default and regarded as required.

- Wind Model: Choose between symmetric and asymmetric definition of your storm. The wind model will determine which fields are displayed and which are hidden (by default) in the node attributes dialog.
  - Symmetric is the basic and default choice, and assumes a simple storm definition will suffice.
  - Asymmetric gives you a lot more options for defining the storm's shape and orientation.
  - PBL (Planetary Boundary Layer) model isn't currently available.
- Basin and Subregion: For informational purposes, these define where in the world the storm is taking place.
- Annual cyclone number: Does not affect calculations, but is valuable for book keeping.

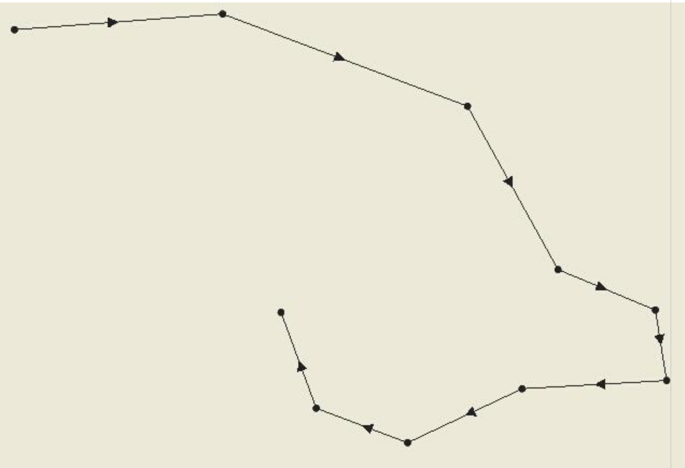
Building a Storm Path

The storm path should be a single continuous line with no breaks or branches. Simply clicking out an arc is sufficient -- vertices will be converted to nodes once you enter the Node Attributes dialog. Do:

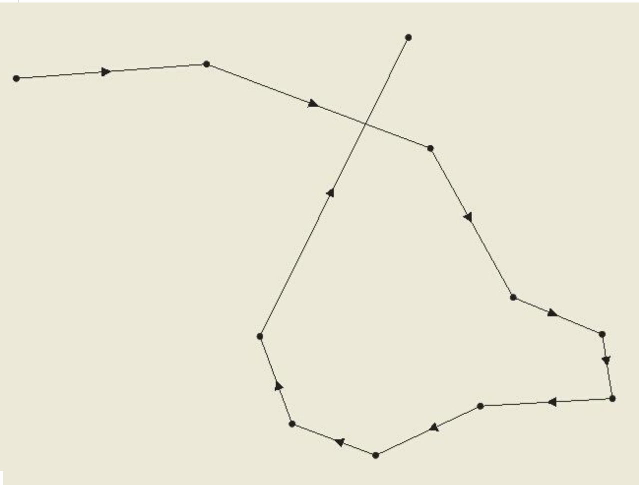
- Operate in Geographic Coordinates
- Make a single path with no breaks
- Create the number of vertices/nodes that you have data for (you can adjust their positions manually with the nodes/vertices selected, or you can set their positions within the Node Attributes dialog)

Don't:

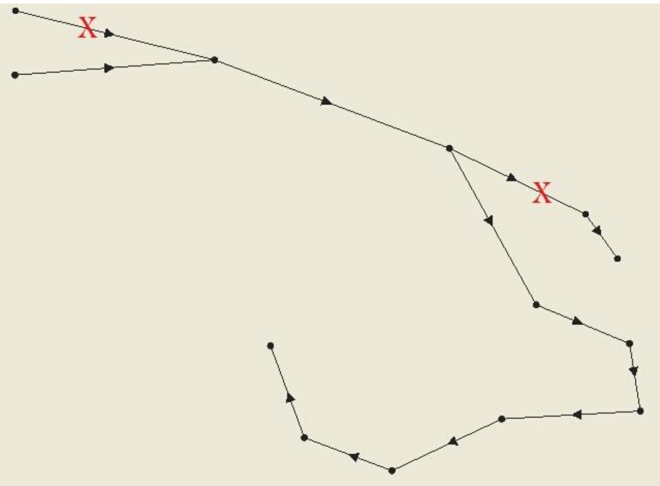
- Create multiple paths in the same coverage
- Split the path in multiple directions
- Create loops with the path



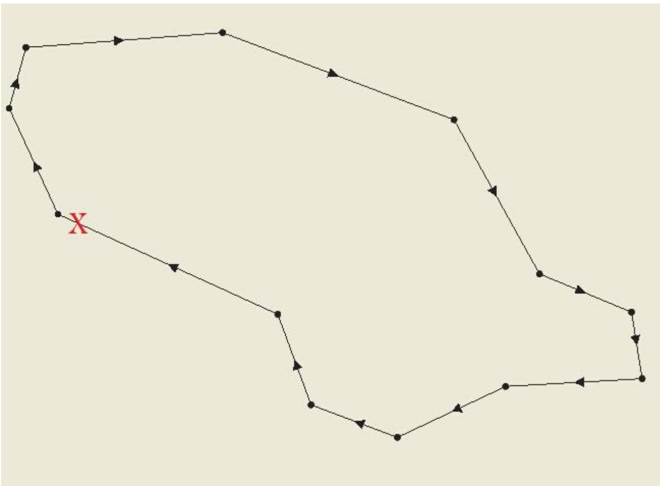
Good example of a storm path



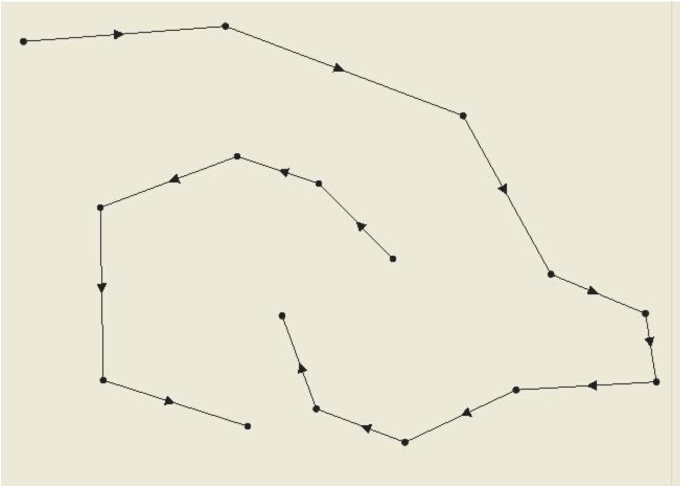
A good path can cross itself as long as it does not connect at the intersection



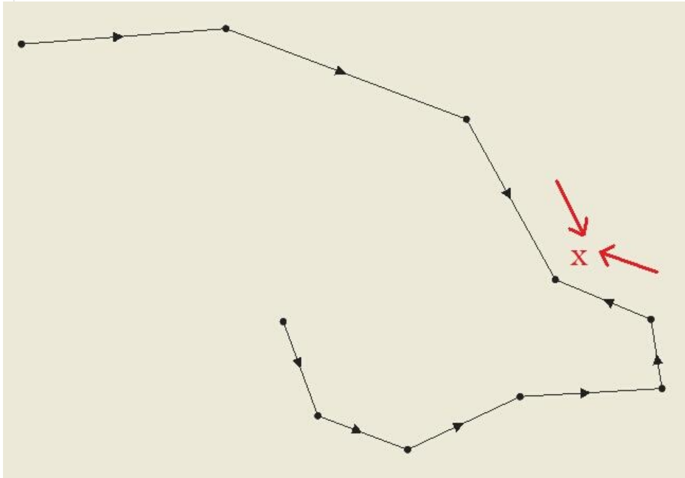
Path should not branch as in this picture



Path should never loop as shown



Only one path can exist. Multiple paths as shown are not allowed.



Different portions of the path cannot converge on each other. The arrows for each section of the path should all point in the same direction for the whole length of the path.

Notes

Consequently, the storm path and nodes information may be obtained in a hurdat file obtained on the NOAA website [1]. Access website and save the Easy to Read version of the file. Extension for file must be saved as hurdat in order to open in SMS.

Node Attributes

Once you've built a storm path, you can define the storm's attributes at each node of the path. Enter the node attributes dialog by selecting the Select Feature Point tool and double-clicking anywhere in the coverage. This opens the Storm Track Node Attributes dialog. Whenever you open this dialog, all vertices in the storm path are converted to nodes automatically.

A second way to open the Node Attributes dialog is to use the Select Feature Point tool to select one or more nodes on the path. Then right-click and select "Node Attributes...". This will highlight those nodes in the dialog.

Fields colored blue and displayed by default are those that are required for the Wind Model (symmetric or asymmetric) selected in the Coverage Attributes dialog. The "Show all"/"Show only required" button can be used to show all the fields available, even those not used by ADCIRC or for the selected wind model. These are useful for book keeping and completeness, even though they typically do not affect calculations.

The Storm start time sets the starting time for the first node in the storm's path. Each node then defines an offset from this starting time in hours (see below). Year, month, day and hour are important, while minutes and seconds should be left at 0.

The fields in the spreadsheet more or less correspond directly to a field in the fort.22 file [2]:

- Lat and Lon: These define the latitude and longitude of the given node, in tenths of degrees (900 = 90 degrees). You can edit these values directly from the dialog, or you can select the nodes with the Select Feature Point tool and edit their X and Y that way. In this dialog, values are always positive and N/S/E/W determines quadrant, whereas the main SMS interface uses negative numbers for South and East.
- TechNum/Minutes (TECHNUM/MM)
- Technique (TECH): ADCIRC recommends that this be set to ASYM when dealing with asymmetric storms
- Time offset (YYYYMMDDHH and TAU): This field combined with the Storm start time above the spreadsheet determine the YYYYMMDDHH and TAU fields in the fort.22. This field is the offset (in hours) from the storm

start time.

- Max sust wind spd (VMAX)
- Minimum sea lvl pressure (MSLP)
- Lvl of tc development (TY)
- Wind radius code (WINDCODE): ADCIRC requires that this be full circle for symmetric, and northeast quadrant for asymmetric. This is fairly restrictive, but SMS can convert many of the other options to northeast quadrant automatically. A model check will warn you if some of your selections cannot be converted without losing data.
- Wind Intensity (RAD, RAD1-4): Each node can store wind intensity and radii for the storm shape at 34, 50, 64 and 100 kts.
- Pressure of last closed isobar (RADP)
- Radius of last closed isobar (RRP)
- Radius of max winds (MRD)
- Gusts (GUSTS)
- Eye diameter (EYE)
- Max seas (MAXSEAS)
- Forcaster's initials (INITIALS)
- Storm direction (DIR)
- Storm speed (SPEED)
- Storm name (STORMNAME)
- System depth (DEPTH)
- Wave height for radii (SEAS)
- Seas radius code (SEASCODE)
- Wave height radius 1-4 (SEAS1-4)

Linking the Coverage to the ADCIRC Project

Once you've defined your storm path and all its data, it's time to link the project into ADCIRC. To do this, select the ADCIRC mesh and go to ADCIRC | Model Control. Choose the Wind tab. Select either Dynamic Holland Model (NWS=8) or Asymmetric vortex, Holland gradient wind model (NWS=9). In the Wind File Options section, click Choose coverage... to select your coverage and link it in. The Options button will now open the Coverage Attributes dialog, allowing you to edit it quickly from this dialog. In the Timing tab be sure to set up the simulation start time and how long it runs. The wind coverage's time span should have the same start time and duration, or be longer so that it encompasses the simulation time span. Also set up any other ADCIRC settings as needed. When finished, click OK, then go to File | Save ADCIRC, then ADCIRC | Run ADCIRC. The model check will alert you to any potential problems before ADCIRC runs.

References

[1] http://www.aoml.noaa.gov/hrd/hurdat/Data_Storm.html

ADH

The ADH Vessel Coverage is used to add vessels to the simulation and give them paths to follow. Generally, one coverage represents one vessel. You can drag one or more vessel coverages into the ADH Mesh to add those coverages to the simulation (by creating links). This allows you to have more vessels defined than you are using in your simulation, and you can swap them in and out of the simulation to run different tests with different vessels.

Boat Path

Each vessel coverage has at least one arc to define the boat path. The arc determines the vessel's starting position and starting speed, and where it will go from there. Additional segments can be added to the arc by adding vertices. Vertices add destinations but do not affect speed. To change the speed of the boat you can convert a vertice into a node, and enter the node's attributes.

In general, there will always be exactly one path per vessel coverage. The path can be made up of multiple arcs, but the arcs should not split into multiple paths. The one exception to the rule of "one path per coverage" is when you have multiple vessels that are identical except for their speeds & paths. In this case you can have multiple separate paths in the same coverage, and each will create its own separate copy of the boat defined in the coverage properties. These paths can cross each other as long as they do not connect to each other (at a node).

Another option is to simply duplicate the coverage after the boat properties are defined. This allows you to add and remove the boats from the simulation separately, and the boats can have vertices and nodes in the same place (same x and y coord) along their path without conflicting.

The boat path writes the FDEF and SDEF cards to the boat file.

Dialogs

Node Attributes

Nodes on arcs change the speed of the boat or, in the case of the first node of the path, define the boat's starting speed. To change the speed of the boat at a node, select it and right-click, then select 'Node Attributes...'. Alternately, you can double-click the node.

Boat Parameters

The boat's parameters are defined in the coverage properties. Right-click on the coverage and choose 'Properties...'. This dialog lets you set the boat's size and shape, and define propellers if desired. If the OP BTS card is included, each vessel in the simulation will need to have propellers defined. Without the OP BTS card, propellers supposedly do nothing.

Each field corresponds directly to a card in the boat file.

- Length (BLEN)
- Width (BWID)
- Bow to Length Ratio (PBOW)
- Stern to Length Ratio (PSTR)
- Draft (DRFT)
- Fraction of Draft Applied to PBOW (CBOW)
- Fraction of Draft Applied to PSTR (CSTR)

Propeller (PROP card):

- Propeller Type: Open wheel or Kort nozzle

- Propeller Diameter
- Distance between propellers
- Tow boat length: This length provides an offset distance of the propeller induced shear stresses from the vessel. Set to 0 if there is no tow boat.
- Distance from prop to tow boat stern

Dredging

The Dredging Operations and Environmental Research (DOER)group in the U.S. Army Corps of Engineers has sponsored a set of dredging tools in the SMS interface. These tools simplify the process of creating numerical simulations of a dredging process. The toolbox actually includes several dredging models of varying complexity. The SMS also includes interfaces for setting up individual simulations using each of these models.

Using this toolbox, an engineer may evaluate conditions at a dredging site, a placement site or both. Equipment used in the project may be selected from a list of existing vessels or defined locally. A typical project consists of the following steps

Flow of Control at Dredge Location

1. Define a new Dredging Simulation. You do this by right clicking in the Project Explorer and selecting "New → Dredging Simulation". This will create a "Dredge" entry in the Project Explorer as well as a new simulation. You will now add specific dredging processes to the simulation.
2. Define the sediment (material types) and **Sediment Parameters**. Right clicking on the "Dredge" entry of the project explorer allows access to these parameters.
3. Define a *Dredging Coverage* for each dredge operating in the project area. In that coverage, specify the equipment (coverage properties), the zone that equipment will work in (polygons with material types) and the paths (or typical paths) it will follow. Drag the dredging coverages into the simulation created in the previous step.
4. Generate PTM sources at the dredging site using specified losses or from results of the dredge source model by right clicking on the simulation and selecting "Generate PTM Sources ...".
5. Run PTM to see where sediment migrates and deposits.

Flow of Control at Placement Location

1. Set up Dredge Location information as specified above.
2. Define a Fate domain as the placement zone. Drag this grid into the dredging simulation.
3. Generate MPFate loads or MKPlacement locations from results of the simulation by right clicking on the simulation and selecting "Generate MPFate ..." or "Generate MKPlacements".
4. If desired, run MPFate in the placement zone to evaluate filling of the placement zone.
5. If desired, run STFate at specific placements in the placement zone to evaluate water quality and produce PTM sources.
6. If desired, run PTM to see where sediment from the placement zone migrates and settles.

Flow of Control for Both Dredge and Placement Locations

1. Set up Dredge Location information as specified above.
2. Set up Placement Location information and run STFate as specified above.
3. Run PTM to see where combined sediments from both the dredge and placement zones migrate and settle.

Additional simulations may be created by right clicking on the "Simulations" item in the project explorer. These may be used to evaluate other combinations or alternatives. Dredging coverages may be in multiple simulations.

Models included in the Dredging Toolbox

- Dredge Source Model – calculates the sediment sources generated at the dredge site. These sources can be input to PTM.
- STFate – models the descent and collapse of a sediment cloud released from a barge or hopper. Outputs concentrations at various depths throughout the process, deposition on the bottom and particle sources stripped out of the cloud. These stripped quantities can be input to PTM.
- MPFate – models the combined effect of multiple STFate placements. Combines the short term placement operation with avalanching and consolidation to evaluate the net effects on the placement zone.
- MKPlacement – computes placement locations over a placement zone. This is one part of MPFate.
- PTM – tracks sediment material moving through modeling domain. Simulates motion of this material as individual particles.
- LTFate – model built on the EFDC/SED-ZLJ engine to simulate long term bed transport. Could be applied over placement zone results from MPFate to simulate later movement of materials away from initial placement.

EFDC

The EFDC coverage is used to create curvilinear grids for use with the EFDC model (EFDC grid format). A curvilinear grid has I and J coordinates but the cells do not have to follow straight lines like rectilinear grids see Curvilinear Grid Module.

The coverage doesn't contain any point or arc attributes.

The EFDC coverages uses the same meshing tools as other SMS coverages but with limited options. The only supported meshing option is rectangular patches. Patches cannot have transitions (all opposing sides must have the same number of vertices). See Mesh Generation for more information about the meshing options available in SMS.

EFDC Model Control

The EFDC Model Control is accessed using the *EFDC | Model Control...* menu item.

Dialog Description

The EFDC Model control contains the following pages:

General

The general tab of the EFDC model control is used to specify general simulation parameters. It contains the following controls:

Title

Text description up to 80 characters in length for the run. EFDC.inp card 1.1

Wetting and drying

- **Method –** Wetting and drying method. EFDC.inp card 5.5
  - **None**
  - **Constant Depth**
    - **Nonlinear iteration**
    - **Cell masking**
  - **Variable Depth**
    - **Cell masking**
    - **Cell faces**
- **Dry depth –** Depth at which cell or flow face becomes dry. EFDC.inp card 11.10
- **Depth adj –** Adjustment to bottom bed elevation field in meters. EFDC.inp card 11.12
- **Layer const –** Log bdry layer const or variable rough height adj in meters. EFDC.inp card 11.5

Coriolis

- **Latitude –** The coriolis parameter will be calculated by SMS from the specified latitude value. EFDC.inp card 8.4
- **Constant –** Constant coriolis parameter in 1/sec. EFDC.inp card 8.4

Solver

- **Residual –** Target square residual of iterative solution scheme. EFDC.inp card 3.2

Time

The time tab of the EFDC model control is used to specify simulation time parameters. It contains the following controls:

Simulation

- **Start time –** Time origin of the run. EFDC.inp card 8.2
- **Time period –** Reference time period in seconds. EFDC.inp card 8.3
- **Num. steps –** Number of time steps per reference time period. EFDC.inp card 7.2
- **Num. periods –** Number of reference time periods in run. EFDC.inp card 7.1

Geometry

The geometry tab of the EFDC model control is used to specify geometry related parameters for the simulation. It contains the following controls:

Vertical Layers

The number of layers specified in the vertical layers spreadsheet is used for the value of EFDC.inp card 9.1.

- **Layer –** Layer number. Use the New and Delete buttons to create / remove layers. EFDC.inp card 10.1
- **Percent –** Dimensionless layer thickness. Must sum to 100%. EFDC.inp card 10.2

Output

The output tab of the EFDC model control is used to specify the output the model will create. It contains the following controls:

Output Files

- **Restart file**
  - **End of run** – Restart file written at the end of the run. EFDC.inp card 2.2
  - **Every <> time steps** – Restart file written every <> time periods. EFDC.inp card 2.2
- **Diagnostic files** – Write diagnostics files for external mode solver. EFDC.inp card 3.10

Advanced Cards

The Advanced Cards tab of the EFDC model control is used to modify advanced simulation parameters not supported directly in the Model Control.

Dialog Description

The text blocks associated with a given card can be edited directly in the dialog. A list of the cards available to edit is shown on the left of the dialog. The fields corresponding to the card parameters directly supported by SMS will be overwritten when the dialog is exited.

ESMF – Earth System Modeling Framework

The Earth System Modeling Framework (ESMF) [1] is used to couple the following Models:

- ADCIRC – STWAVE

To create a coupled simulation using ESMF, perform the following steps:

- Right click in the Project Explorer
- Choose the menu *New Simulation | Hurricane*
- Change the simulation name if desired
- Create the elements to be include in the simulation and drag tree pointers representing them into the simulation. The following items can be linked to an Hurricane simulation:
- ADCIRC – STWAVE
  - ADCIRC Mesh
  - STWAVE Grid
  - Hurricane Coverage
  - PBL Wind Coverage
  - WAM Simulation
- Right click on the Hurricane simulation in the Project Explorer
- Choose the menu Properties. This action will create a new Hurricane coverage containing polygons that identify overlapping grid sections and areas of interest defined by the user. It will then bring up a *Hurricane Mode Project Summary* property sheet containing four tabbed dialogs.
  - Overview
  - ESMF
  - Spatial View
  - Timeline
- Right click on the ESMF simulation and choose Export ESMF Files. SMS will create a folder named ESMF in the same directory as the .sms file of the current project. Inside of the ESMF folder will be a folder and input files for the ESMF Simulation. The ESMF Simulation folder name will be the same as the name given to the ESMF Simulations in the *Project Explorer*.

ESMF Hurricane Overview

The dialog gives an overview of the components of the Hurricane simulation. It also allows the user to designate the number of processors to be use by each model in the simulation.

- Simulation Name – Name of the Hurricane simulation as set in the project explore
- Properties – Properties of the Hurricane simulation
  - Model Name – Name of models found in the simulation
    - ADCIRC <mesh name> (1 required) – Name of the adcirc mesh used in the hurricane simulation.
    - STWAVE <grid name> (1 required) – Name of the stwave cgrid used in the hurricane simulation.
    - PBL <pbl cov name> (optional) – Name of pbl wind coverage used in the hurricane simulation.
    - WAM <wam sim name> (optional) – Name of wam simulation used in the hurricane simulation.
- Projection – Projection type of model.
  - Geo – Geographic (Latitude/Longitude)
  - STPL <#> – State Plane number
  - Other – Other type of projection
- Start Time – Temporal starting time of model.
- End Time – Temporal ending time of model.
- Threads – Number of processor threads used by model for computation or I/O processing.
- Function – Function used to adjust the processor threads used by each model.
  - Set Threads – Set the number of processor threads for:
    - Computational or I/O processing (ADCIRC)
    - Grid Partition I and J processing (STWAVE)
  - NONE – WAM and PBL use only one processor thread.

Hurricane ESMF

- Set the ESMF simulation options in the *Data Exchange* and *Area Mapping* sections of the dialog
  - Data Exchange
    - Model A – The meshed based model to be use (hard-coded to ADCIRC)
    - Data Exchange – Controls how data is exchanged between the two models (↔ bi-directional, → uni-direction A to B, ← uni-directional B to A)
    - Model B – The grid based model to be use (hard-coded to STWAVE)
    - Model A → B – How frequently the results of Model A are passed to Model B
    - Units – Frequency units for A to B exchange (days/hours/minutes/seconds)
    - Model A ← B – How frequently the results of Model B are passed to Model A
    - Units – Frequency units for B to A exchange (days/hours/minutes/seconds)
- Area Mapping
  - ID – Polygon identifier correlated to the ids visible in the main graphics window.
  - Mapping
    - Single – Model A exchanges data with a single instance of Model B
    - Combined – Model A exchanges data with multiple instances of Model B. Model A can receive either the average or maximum values from the instances of Model B.
  - Option – If the "Mapping" field is set to "Single" and multiple grids overlap the identified polygon, this field allows the user to select which grid will be used. If the "Mapping" field is set to "Combined", this field allows the user to select how the data from the grids will be combined (Average or Maximum.)

Hurricane Spatial View

This dialog provides a view of any grid boundaries (WAM and STWAVE), grid frames (PBL), and the ADCIRC mesh boundary associated with the simulation.

Hurricane Timeline

This dialog displays the timelines for each simulation (WAM, STWAVE, PBL, and ADCIRC) that are part of the hurricane simulation. The start and end times need to be set for each simulation individually.

Related Topics

- Steering

References

[1] http://www.esmf.ucar.edu/

FATE

Graphical Interface

The interface can be broken down into tools and options for three categories. These categories include the placement region, equipment, and dredged material attributes. The options that comprise each of the categories are listed below.

Placement region attributes:

- Location, resolution, size and orientation of the placement grid.
- Ambient currents and bathymetric properties.
- Location of dumping site.
- Local water densities.

Equipment attributes:

- Vessel size and draft.
- Bin size.
- Vessel speed and direction.

Dredged material attributes:

- Composition (percentages of materials, particle sizes and velocities, void ratios, etc.).
- Material density.
- Volume of material load.
- Contaminant concentration level.

FATE Models

The FATE interface can be used with two similar models that both deal with the placement of dredged materials.

STFate

Models water quality/sediment concentrations for single placements for regulatory compliance.

MPFate

Models multiple placements to predict final bathymetry in a placement area.

MK Placement

Assists the user in developing placement scenarios for MPFATE.

FATE Model Menu

The FATE model menu can be accessed by creating a placement region and activating the Cartesian Grid module. The following are summaries of the available accessible options from this menu.

- **Model Control –** The following table describes the different tabs of the Model Control dialog and the options available on each tab

Tab	Functionality
General	Specify run type, simulation time and time step length, bottom roughness coefficient
Coefficients	Define simulation coefficients (usually the default values are sufficient)
Water Density	Enter water density values for the entire depth of the placement region (densities can be calculated using salinity and temperature input)
Tracer/Constituent	Describe a constituent (or contaminant) and its properties
Output	Set times and depths at which results should be written for
Current	Define direction and magnitudes of velocities of the placement region currents

- **Placements –** Give the location, vessel direction and speed, and time for placement operation.
- **Loading –** Define the layers of placement materials and sediment composition of each layer.
- **Vessel –** Select or define the vessel being used in the operation.
- **Sediments –** Provide attributes of the various sediments which make up the placement material.
- **Run Fate –** Launches the STFate model using the active simulation input files.

All input data found in these dialogs and tabs are summarized by the STFate model in the *.due file created at the beginning of a successful simulation run. The *.due file is the standard input file for the STFate model executable.

Placement Regions/Disposal Sites

A placement region or disposal site is created in two steps. First, the Map Module is used to create the domain of the desired region as a 2-D grid frame. Then, the domain is mapped to a 2D cartesian grid so the attributes of the placement region can be specified.

How to Create a Placement Region

To create the disposal site or placement region, first, make sure the Map Module is active. Now, create a new coverage in the Project Explorer. Change the coverage type to FATE by right-clicking on the coverage and selecting FATE from the Type option. From the toolbar, select the 2-D Grid Frame tool and click out a 2D grid frame. Next, select Feature Objects | Map → 2D Grid. The Map → 2D Grid dialog will appear. Here, the size and angle of the grid can be changed along with the bathymetric and current properties associated with the region.

Compliance Reports

SMS can output compliance reports for STFate simulation runs. These reports specify whether the material concentrations are below a specified threshold. For more information visit STFate Compliance Reports.

GenCade

The GenCade coverage allows the user to define the geographical location of the characteristics and structures of a GenCade simulation. One or more coverages can be mapped to the simulation.

See the GenCade documentation for more information.

Generic Mesh Coverage

Generic 2D mesh coverage

Generic mesh coverage is for a Generic mesh model. The generic model interface spans both the map and the mesh modules. You can create a Generic Model coverage and assign arc boundary conditions based on the types defined in the generic model template. You can also assign attributes to feature points. More information about a meshing coverage can be found under the article Coverages.

Convert Map feature arc and point attributes (Generic 2D mesh coverage) to mesh nodestring and node attributes

Users in the map module with a Generic 2D mesh coverage can create feature points and arcs. These points and arcs can then be assigned attributes. This is done by double clicking on either the feature point or feature arc. For points a dialog will appear allowing you to assign node or element boundary conditions. For arcs you will be assign attributes on the arc. When doing a Map→2D Mesh, point node boundary conditions will be assigned to the nearest node. Point element conditions will be assigned to the nearest element. Mesh nodestrings will be created and assigned from the nodes nearest the feature arc.

Convert a Map to 2D Mesh

To convert a map to a mesh, right click on the map default or active coverage. Then select "Convert" then "Map→2D Mesh". A dialog is displayed. If a mesh already exists, you may choose to delete it, or map the attributes to the existing mesh.

LTFate

LTFate Coverage

The LTFate coverage is used to create curvilinear grids for use with the LTFate model (EFDC grid format). A curvilinear grid has I and J coordinates but the cells do not have to follow straight lines like rectilinear grids see SMS:Curvilinear_Grid_Module.

The coverage doesn't contain any point or arc attributes.

The LTFate coverages uses the same meshing tools as other SMS coverages but with limited options. The only supported meshing option is rectangular patches. Patches cannot have transitions (all opposing sides must have the same number of vertices). See SMS:Mesh Generation for more information about the meshing options available in SMS.

Synthetic Storm Coverage

Background

When simulating an actual storm (hindcast), the storm information such as the track locations, central pressures, radius information, speeds, and Holland B values can come from an analysis on data collected during the storm. However, if our simulation is intended for design analysis you may not want to choose a storm that has happened but a storm which may happen. Often several configurations for storms would be analyzed to see the results of each. The synthetic storm coverage and associated generator executable provide a mechanism for creating a PBL coverage (trop file) based upon user specified parameters.

The first step is to decide upon a track path or multiple paths to simulate. For multiple paths, one option is to create a single track and use the perturbation tools in SMS to generate similar tracks by using offsets and modifications of central pressures, etc.

Once your track locations have been defined, we need to define the associated data for the track (central pressures, Holland B, storm radius, etc.). The US Army Engineer Research and Development Center has developed a utility that creates a full PBL input trop file from a small set of user defined parameters.

Inputs

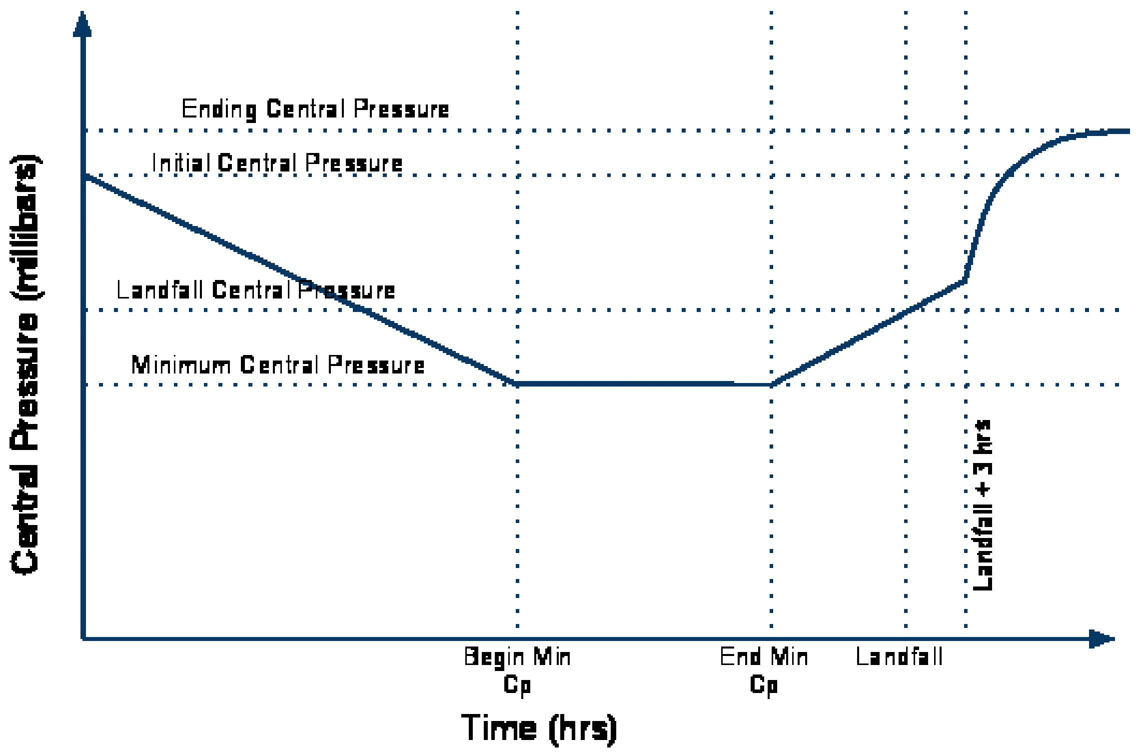
The following parameters are used to fill in the track data:

- Storm name
- Storm number
- Starting date/time (nearest hour)
- Forward speed in knots
- Far field atmospheric pressure in millibars
- Initial central pressure in millibars
- Minimum central pressure in millibars
- Landfall central pressure in millibars
- Initial storm radius in miles
- Landfall storm radius in miles
- Initial Holland B value

In addition to these parameters the locations where minimum central pressure exists along the track and the landfall location are specified.

Methodology

ERDC has developed curves to represent general behavior of storms. These curves are a simplistic approach and should be used with caution. When in doubt if you are providing appropriate wind fields consult your favorite meteorologist. Sample curves for central pressure, Holland B, and storm radius can be seen below. Some of the values shown are user supplied and others are determined by the synthetic storm generator (like ending radius).



User Interface

Synthetic storms are created in SMS using the "synthetic storm coverage." This coverage is in the folder "\Models\Wind" when accessing it from the new coverage dialog or changing the type of an existing coverage.

The coverage must contain a single track made up of multiple arcs. The locations of starting minimum central pressure, ending minimum central pressure, and landfall are specified on arc end nodes. You can change the type of an arc node by selecting the node, right clicking, and choosing the desired type from the "Type" submenu. When you change a node to any type except for generic and a node of that type already exists, the existing node will be changed to a generic node. There can only be one node of each "non-generic" type.

The parameters for the storm generator are stored with the coverage and can be accessed by right-clicking on the coverage and choosing "properties."

When the track has been defined, the node types specified, and the properties assigned to the coverage the PBL coverage can be created. To create the PBL coverage, right-click on the coverage and choose "Create PBL coverage." This will run the ERDC storm generator utility and read the resulting data into a new PBL coverage.

TUFLOW Coverages

TUFLOW models use several types of Feature coverages. Feature coverages are used in SMS to represent geometry and associated attributes as points, arcs, and polygons. Unlike most models used in SMS, TUFLOW uses coverage data as model inputs.

TUFLOW coverages can be associated with 2D Geometry Components and TUFLOW Simulations by creating links to the coverage in the component or simulation. The use of links allows multiple simulations or geometry components to share coverage data. Sharing data between simulations reduces required disk space for the simulation and makes it easier to update several simulations with the same changes.

In addition to the coverages listed below, TUFLOW models can use Area Property Coverages to define where to use specific material properties using polygons.

TUFLOW 1D/2D Connections Coverage

1D/2D connections are used with the 2D BC coverage to link 2D and 1D domains.

Two types of arcs can be created in a 1D/2D connection coverage: 1D/2D connection arcs (CN) and Flow vs. Head Connection arcs (SC).

1D/2D Connection arcs are written to a 2d_bc layer file for TUFLOW input (see 2d_bc_layers in the TUFLOW documentation). One end of the connection arcs must end at the same location as a 1D Flow/2D Water Level Connection (HX) in a TUFLOW BC coverage. The other end of the connection arc must end at the same location as a network node in a TUFLOW network coverage.

TUFLOW 1D Cross Section Coverage

Cross section coverages are used to define open channel cross section data for 1D networks. Cross section geometry is generally extracted from a TIN and may be edited by hand if desired.

Cross sections are created by creating arcs in a cross section coverage. SMS can automatically create cross section arcs from a 1D network.

After cross section arcs are created, the elevations for the arc can be extracted from a scatter set (TIN) by right clicking on the coverage and selecting *Extract from Scatter* from the menu. This will extract the cross-section data from the active dataset on the active scatter set. Values are extracted at each triangle edge in the scatter set and each node or vertex on the cross section arc.

After cross sections have been created, the materials from an area property coverage may be mapped to the cross-sections. This is done by right clicking on the coverage and selecting, *Map Materials from Area Property Coverage*.

Individual Cross sections may be viewed and edited by double clicking on an arc, or by selecting an arc and choosing Feature Objects→Attributes. This will bring up the cross-section attributes dialog (see TUFLOW Cross Section Attributes).

All the cross-sections in the coverage are stored in a cross section database. The cross section database may be opened by right clicking on the coverage and selecting *Properties* (see Editing Cross Sections).

TUFLOW 1D Network Coverage

1D domains are made up of a network of channels and nodes. Channels represent the conveyance of the flowpaths and nodes represent the storage of inundated areas (TUFLOW Users Manual). Channels are created using arcs and the arc endpoints are the nodes.

There are a variety of channel types including open channels, weirs, and culverts (pipe networks). The channel also has a variety of attributes depending upon the channel type. The channel type and attributes are defined in the *Channel Attributes* dialog.

There are two types of nodes generic nodes, and inlets. Generic nodes can be used to specify storage and can be used to set channel invert elevations. Inlet nodes are used to get flows from a 2D domain into a 1D pipe network below the 2D domain. The node type and attributes are defined in the *Network Node Attributes* dialog. The TUFLOW documentation sections 4.5.1.3: Connecting Pits and Nodes to 2D Domains and 4.5.3: 1d_nwk Attributes list some new TUFLOW features that SMS now supports. SMS's Network Node Attributes dialog has several new additions to the Create Connection to 2D Domain (SX) section. These new options allow you to control elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make lookup easy.

The TUFLOW documentation sections 4.5.1.3: Connecting Pits and Nodes to 2D Domains and 4.5.3: 1d_nwk Attributes list some new TUFLOW features that SMS now supports. SMS's Network Node Attributes dialog has several new additions to the Create Connection to 2D Domain (SX) section. These new options allow you to control elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make lookup easy.

TUFLOW 2D/2D Links Coverage

A 2D/2D Links Coverage is used to setup TUFLOW to use multiple 2D domains. See Linking 2D Domains.

TUFLOW 2D Flow Constriction Shape Coverage

A 2D Flow Constriction Shape coverage is used to define flow constrictions in TUFLOW. These are used to model hydraulic structures or other sources of additional losses in TUFLOW. Flow constrictions are of two categories: standard (non-layered) and layered flow constrictions. Layered flow constrictions can be used to model situations where flow has multiple pathways at different elevations. Examples would include flow under a bridge and over the bridge deck as well as a pipeline (typically large box culverts to model as 2D) crossing a waterway. The standard (non-layered) flow constrictions can model box culverts, floating bridge decks, bridges, or apply additional form losses to an area (due to submodel scale features). Flow constrictions can be created for arcs or polygons.

Note that this coverage is different from the older (and outdated) 2D Flow Constriction coverage, which is still supported.

Overview

Standard (non-layered) Flow Constrictions

Flow constrictions introduce additional losses and/or reduced flow areas through an area of the domain. Flow constrictions can be used for large box culverts, bridges, or floating bridge decks. Flow constriction properties include invert elevations (optional), low chord (obvert) elevations for structures, blockage information, manning n values, and form loss coefficients.

Layered Flow Constrictions

Here is the description from the TUFLOW manual:

> Four layers (not GIS layers!) are represented, with the bottom three layers each having their own attributes. The top, fourth, layer assumes the flow is unimpeded (eg. flow over the top of a bridge). Within the same shape, the invert of the bed, and thickness of each layer can vary in 3D. Each layer is assigned its own percentage blockage and form loss coefficient. For example, the layers of a bridge structure would be along the following lines.

- Layer 1: Beneath the bridge deck. Might be 5% blocked due to the bridge piers and have a small form loss for the energy losses associated with the piers.
- Layer 2: The bridge deck. This would be 100% blocked and the form loss coefficient would increase due to the additional energy losses associated with flow surcharging the deck.
- Layer 3: The bridge rails. These might be anything from 100% blocked (solid concrete rails) to 10% blocked (very open rails). Some form losses would be specified depending on the type of rails.
- Layer 4: Flow over the top of the rails - flow assumed to be unimpeded.

> Layered FCs function by adjusting the flow width of the 2D cell so as to represent the combination of blockages of the four layers, and by accumulating the form losses. When the flow is only within Layer 1, only the attributes of Layer 1 are applied. As the water level rises into Layer 2, the influence of the Layer 2 attributes increase as the water continues to rise. Similarly for Layer 3 and Layer 4. The cell side flow width is calculated by summing the flow areas of each layer (including the effects of layer blockages), and dividing by the water depth. The form losses are applied as follows:

- If the water is entirely within Layer 1, the Layer 1 FLC is applied.
- If the water level has reached Layer 2, the value applied is the Layer 1 FLC plus a fraction of the Layer 2 FLC based on the depth of water within Layer 2. For example, if Layer 1 FLC is 0.1 and Layer 2 is 0.5, and the

water is 40% of the way up Layer 2, the FLC applied is 0.1 + 0.4*0.5 = 0.3.

- Similarly, if the water level is into Layer 3, the FLC is the Layer 1 FLC plus the Layer 2 FLC plus a fraction of the Layer 3 FLC.
- Once the water level is above Layer 3, ie. is into Layer 4, the FLC is held constant at the sum of the FLCs for Layers 1 to 3.

--TUFLOW Manual 2008-08

Varying Elevations Within Flow Constriction

Flow constrictions can be specified as polygons or arcs (along a line). For a flow constriction defined by a polygon, some values may be varied within the polygon by using arc and point attributes. For standard flow constrictions, the invert and low chord (obvert) elevations can be modified within the polygon. For layered flow constrictions, the layer elevations, blockage, form losses, and invert elevations can vary within the polygon.

In order to distribute these vertices for the flow constriction, TUFLOW creates a TIN from the specified data points and interpolates these values through the flow constriction. The flow constriction dialog has several attributes which influence how this TIN is generated and applied. The final elevations for the grid can be verified by reading the "zsh_zpt_check.mif" file into the GIS coverage. The final information about standard flow constrictions can be verified using check files. The "fcsh_uvpt_check.mif" for standard flow constrictions and "lfcsh_uvpt_check.mif" for layered flow constrictions (see SMS:TUFLOW Check Files).

Arcs on the perimeter or within a polygon flow constriction can be used to spatially distribute the properties above. To use an arc for elevations, you need to specify it to be of type "Breakline Elevations" in the arc properties dialog. Elevations can be specified at nodes at the end of arcs but not on intermediate vertices. Therefore, in order to represent curved flow constrictions or changes in slope, you need to convert vertices to nodes to create multiple arc segments. These arcs will become breaklines in the TIN generated by TUFLOW. Points within a polygon can also be used to specify elevations and will become an individual point in the generated TIN.

It is important to keep in mind when using flow constrictions that elevation data comes from a combination of the cell elevations, elevations specified with the flow constriction itself, and elevations assigned to perimeter nodes or interior points depending upon the whether each kind of data exists and the options chosen for the flow constriction.

Creating Flow Constrictions

To create a flow constriction for an arc or polygon:

- Create a 2D Flow Constriction Shape coverage and an arc/polygon in the coverage. Note that the old 2D Flow Constriction coverage is also still available, and is different from the one we are creating now.
- Double click on the arc/polygon and change the type to Flow Constriction or Layered Flow Constriction and set the options according to the guidelines below.

Flow Constriction Options

Breakline Elevations

- **Automatically Inserted Vertices** - Used to affect how TUFLOW generates the TIN to use to interpolate elevation data within the flow constriction. Additional vertices can be useful to provide a better triangulation and more smooth transitions between values. The default is for TUFLOW to generate additional vertices so that the final spacing is not more than half the cell size.

Flow Constriction Options

- **Width –**
- **Override Invert Elevation –** Unless overriden, TUFLOW will use the 2D cell elevations for the invert elevations of the flow constriction.
- **Option –** This controls when invert elevations will be applied. If the choice is Minimum, the new invert elevations will be used only if they are lower than the original cell elevations. Similarly, if the choice is Maximum the new elevations will only be used if they are higher than the original cell elevations. If All is chosen, the new elevations will be used regardless if the elevations are higher or lower.

TUFLOW has the option to merge elevations at the perimeter of the flow constriction with the existing elevations that exist in the perimeter cells to make the transition between the elevations smooth. The "Default" option (polygons only) will merge elevations of perimeter points that do not have a specified elevation value but apply specified elevations at points. The "All" option (arcs only) will ignore specified elevations at perimeter points and merge the values with the perimeter points. The "No Merge" option will ignore existing cell elevations and the elevations will come from specified node values if they exist otherwise they will come from the elevations specified for the flow constriction.

Layered Flow Constriction Options

- **Invert Elevation Offset –** This will offset the invert elevations whether they are specified or are the original cell elevations.
- **Layer 1 Low chord –** The low chord (obvert) represents the top of layer 1. For a bridge this is the low chord of the bridge deck.
- **Layer 2 and 3 Depths –** This represent the depth of the layers after layer 1. The obvert of each layer is determined on a cell by cell level from the layer1 low chord/obvert elevation.
- **Blockage % –** This represents the percentage of the flow width lost due to large piers, railings, or other flow impediments. If half of the flow is blocked, the percentage lost is 50%.
- **Form Loss –** The form loss is the ratio of the dynamic head that is lost in the structure. This value can be larger than 1.0 since all of the energy does not have to come from the velocity component. This can be estimated from experience or by comparing with other numeric codes.

Non-Layered Flow Constriction Options

- **FC Type –** Flow Constriction Type. Options are:
  - **General –** Does not include allowances for any vertical walls or friction from underside of deck.
  - **BC –** Box Culvert
  - **BD –** Bridge Deck
  - **FD –** Floating Deck
- **Low chord (obvert)/BC Height/Bridge Deck Depth –** The usage of this field changes depending on FC Type (above). Enter a sufficiently high value (eg. 99999) if there is no obvert constriction.
  - **General** or **BD –** Low chord/Obvert (soffit) of constriction in m above datum.

  - **BC –** The height of the box culvert.
  - **FD –** Floating depth (m) of the deck (ie. depth below the water line).
- **BC Width –** The width of one BC culvert barrel in metres. For example, if there are 10 by 1.8m wide culverts, enter a value of 1.8.
- **Blockage % –** The percentage blockage of the cells. For example, if 40 is entered (ie. 40%), the cell sides are reduced in flow width by 40%, ie. is set to 0.6 times the full flow width.
- **Manning's n –** According to TUFLOW's docs: *For box culverts (BC), the Manning's n of the culverts (typically 0.011 to 0.015) should be specified. This value prevails over any other bed resistance values irrespective of where in the .tgc file they occur (the exception is if another FC BC object overrides this one). If set to less than 0.001, a default value of 0.013 is used.*

For bridge decks (BD), can be used to introduce additional flow resistance once the upstream water level reaches the bridge deck low chord/obvert (or soffit). For floating decks (FD) this is always the case as the deck soffit is permanently submerged. The additional flow resistance is modelled as an increase in bed resistance by increasing the wetted perimeter at the cell mid-side by a factor equal to (2.*Bed_n)/FC_n. For example, if the FC Manning's n and the bed Manning's n values are the same, the wetted perimeter is doubled, thereby reducing the conveyance and increasing the resistance to flow. To be used as a calibration parameter to fine-tune the energy losses across a bridge or floating structure. Ignored for General FC Types.

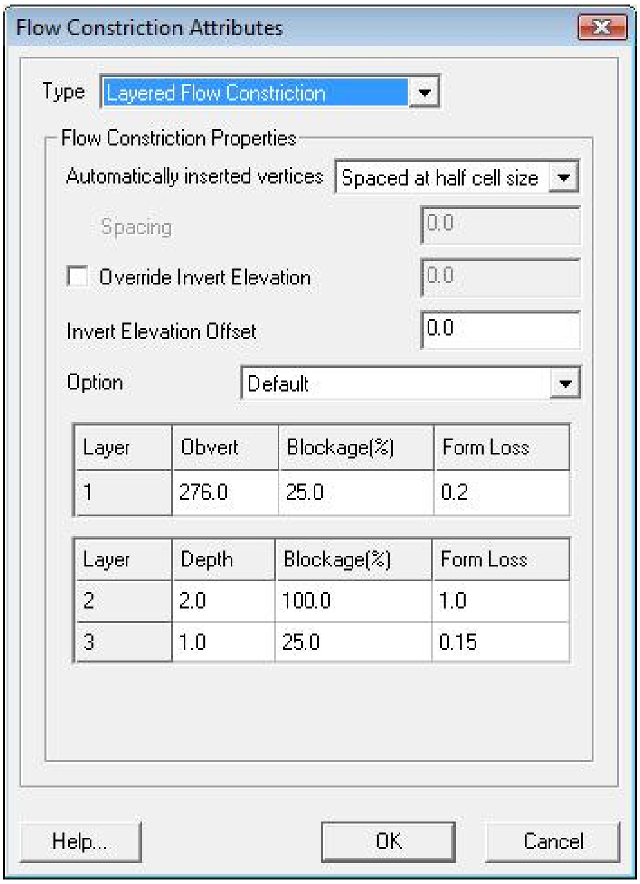
- **Form Loss Coefficient –** Form loss coefficient to be applied above and below the FC low chord/obvert. Used for modelling fine-scale "micro" contraction/expansion losses not picked up by the change in the 2D domain's velocity patterns (eg. bridge pier losses, vena-contracta losses, 3rd (vertical) dimension etc). The effect of these fields changes between arcs and polygons, thick and thin lines, as detailed in the TUFLOW docs (section 4.7.2).

Node Options

All nodes have the option of overriding the invert height at their location. Beyond this, the fields available in the node attributes dialog change based on whether the node is inside (or on the edge) of a Layered Flow Constriction Polygon. If so, the node properties will resemble that of the layered polygon's with layers 1–3. Otherwise only the Low chord (obvert) can be modified.

As with polygons, the Low chord can actually mean different things depending on the settings of FC Type and other factors. This is determined by the polygon the node is on or inside of.

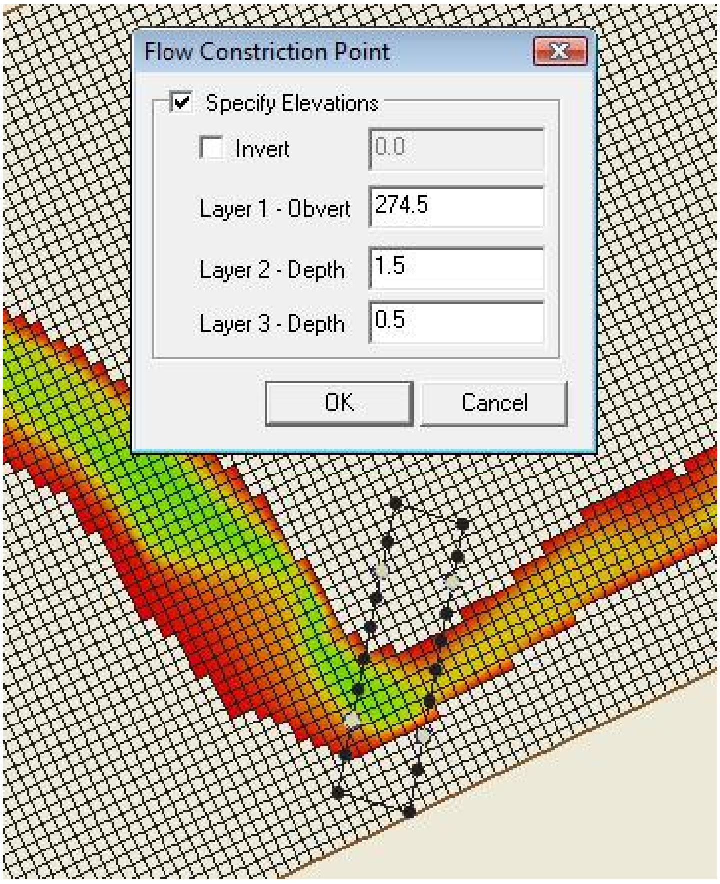
Example: Arched Bridge



Flow Constriction Attributes Dialog

Because it's hard to glean applicable knowledge from a dry description of dialogs, here's an example of how they might be used. We're going to create an arched bridge using a Flow Constriction polygon and Breakline Elevations.

1. Start by creating your Flow Constriction Shape coverage.
2. Next create 4 connected arcs in a rectangle for the polygon for your bridge. Go to Feature Objects->Build polygons to create the polygon.
3. Select the polygon and enter its attributes dialog. Fill the values as follows:
4. Click the Select Feature Arc tool and select both arcs along the sides of the bridge. We want to turn both these arcs into breakline elevations. Right-click and select "Attributes...".
    1. Set the Type to Breakline Elevations (TIN)
    2. Set Automatically insert vertices to "Spaced at half cell size".
5. Next we want to create some nodes along these arcs so we can set different low chord elevations at each one to create the arch. With the arcs still selected, right-click and select "Redistribute Vertices...". In the dialog, choose Number of Segments for the Specify field and enter 9 as the number of segments. Click OK. This splits up the lines so that there are a total of 8 vertices on each, which we will convert to nodes.
6. Choose the Select Feature Vertex option and select all the vertices on the bridge sides (either with a drag-select or by holding shift). Right-click and choose Convert to Nodes.
7. Now it's time to set the node data. Choose the Select Feature Point tool. For each node along the sides of the bridge, enter its attributes dialog (right-click with it selected and choose "Node Attributes...") and insert the data below. You can multiselect nodes with the shift key and edit their attributes simultaneously. Since both sides of the bridge will be the same, and the data below is symmetrical on each end of the bridge, you can select 4 nodes at once to set their values to save time.

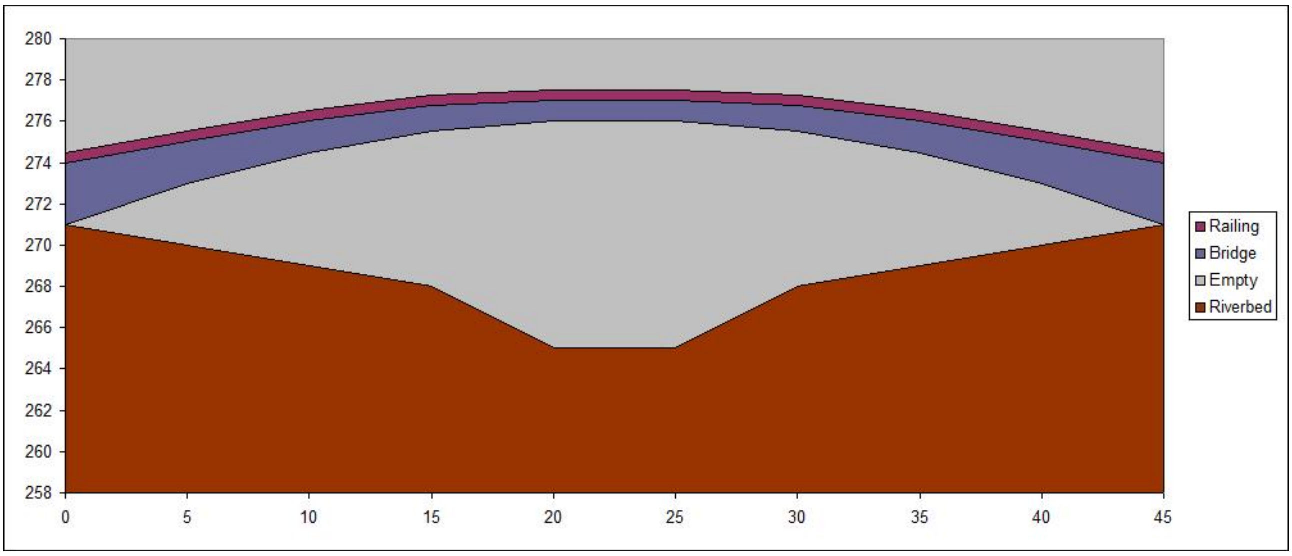


Flow Constriction Point Dialog

Node	L1 Low chord	L2 Depth	L3 Depth
Node1	271	3	0.5
Node2	273	2	0.5
Node3	274.5	1.5	0.5
Node4	275.5	1.25	0.5
Node5	276	1	0.5
Node6	276	1	0.5
Node7	275.5	1.25	0.5
Node8	274.5	1.5	0.5
Node9	273	2	0.5
Node10	271	3	0.5

We now have an arched bridge that will look something like the image below from the side. We added 25% blockage under the bridge to cover things like pillars holding the bridge up, vegetation, fairy-tale trolls hiding underneath, and so on. Layer 2 is the bridge itself, which provides 100% blockage because our bridge is solid concrete and does not allow any water to flow through that section. Above that, the railing gives 25% blockage but otherwise lets water through just fine. And above that? Nothing—any water that rises above the railing of the bridge is completely unrestricted.

The varying width of Layer 2 allows us to create a bridge that is thickest on the ends of the bridge and thin in the center. Our Layer 1 blockage takes support pillars into account, and TUFLOW will automatically determine how much of layers 1, 2 and 3 are affecting the water at any given time and calculate the flow constriction from there.



Bridge Layer Definitions

TUFLOW 2D Spatial Attributes Coverage

Spatially varied attributes can be defined using the 2D spatial attributes coverage. The values for the attributes are assigned to polygons. TUFLOW assigns the values from the polygons to the cells that exist within the polygon.

The two available spatial attributes are initial water levels and weir factors.

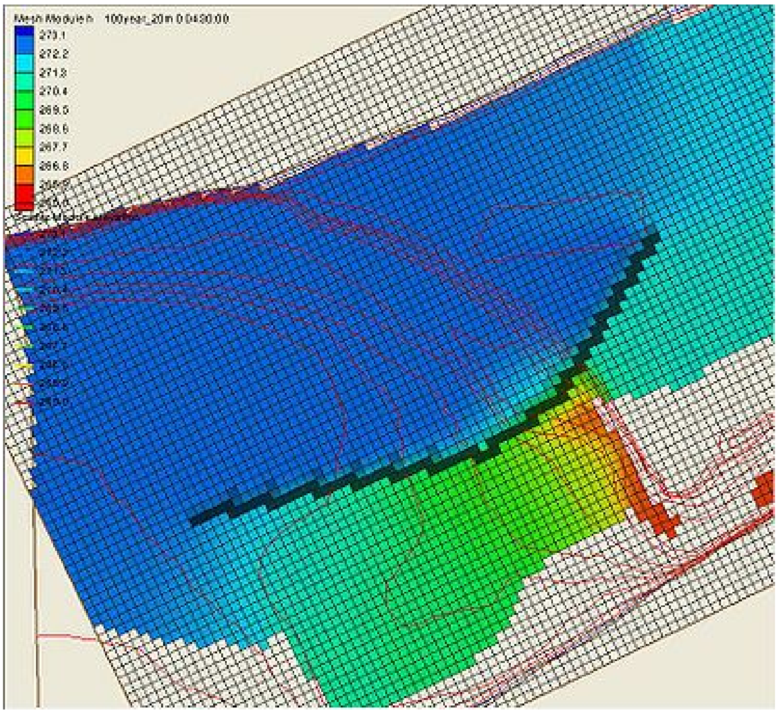
Assigning spatially varied initial water levels may be necessary to ensure that water bodies have water in them initially.

TUFLOW 2D Z Lines (Advanced) Coverage

The 2D Z Lines (Advanced) Coverage is similar to the geometry modification coverage which allows you to modify terrain along an arc or arcs, but has additional options. With the 2D Z Lines (Advanced) coverage, you can modify geometry through time to simulate levee failures or other changes to elevation data within the model run. These changes are set up start when a "trigger" is activated such as at a specific time during the simulation or when water depth exceeds a certain amount. An example application is a levee collapsing when flood water overtops it.

- Make sure the coverage is in the Geometry Component(s) you wish to apply to the modification.

TUFLOW also supports the features described above using polygon ZShapes. However, SMS does not support polygon ZShapes using these advanced features (such as triggers) at this time. If you need to create static modifications to terrain using polygons you can use the 2D Z Lines/Polygons (Simple) Coverage.



Failing Levee

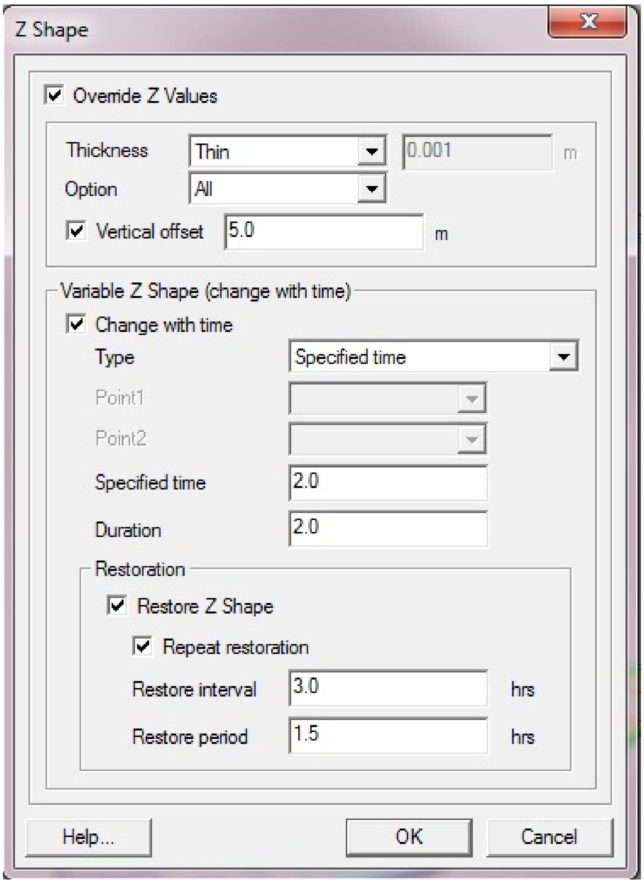
The 2D Z Lines (Advanced) Coverage

The 2D Z Lines (Advanced) coverage is available under the models/TUFLOW folder. ZShape data is stored in a feature arc's attributes. The Z values of the feature arc's points determine its Z, not the Z of the arc itself (exceptions to this are explained in the Arc Properties Dialog section below).

ZShapes are split into two main categories, Static ZShapes and Variable ZShapes. Static ZShapes are simple terrain modifications that do not change over time. They can be used to create a levee, pit, sand bar, dam, etc. Variable ZShapes have a trigger that causes them to raise or lower the terrain during the simulation. Often a variable ZShape is combined with a static one to provide raised terrain for the trigger to modify. This is necessary because a variable z shape cannot raise the same area it intends to lower when its trigger activates -- it needs the terrain to already be there and cannot create it itself, only modify once its trigger activates.

Arc Properties Dialog

Arcs in a 2D Z Lines/Polygons (Advanced) coverage store ZShape data only if they are told to "override Z values". Otherwise they are not exported for TUFLOW's use. To reach the properties dialog,right-click on an arc and choose "Attributes...". Below is an explanation of the zshape options.



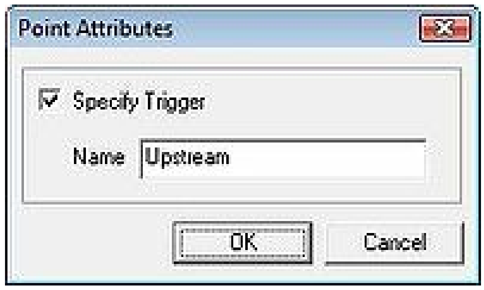
Z Shape Properties Dialog

- Override Z Values: ZShape data is only stored and exported if this checkbox is enabled. It tells TUFLOW to override the current terrain Z values with the new values from the arc.
- Thickness: Arcs can be Thin Lines, Thick Lines, or Wide Lines. Thin lines have a width of 0, thick lines have a thickness less than or equal to 1.5 times the 2D cell size, and wide lines are any width larger than that. Thin lines follow some special rules that affect other options in the dialog, such as Option and Trigger Type.
- Option: The ZShape Options are All, Add, Min, Max, and Offset. Offset is only available for variable thin line Z Shapes, which cannot use the Add, Min or Max options. Add is unavailable for any variable Z Shape.
  - Offset is not listed by the same name in the TUFLOW documentation and is an alias in SMS for not specifying "NO MERGE". This causes the points of the arc to not be written, and the Z of the arc is determined by the Offset field.
- Offset: Raises the entire arc by this amount. Negative values are appropriate.
  - When offset is specified as the Option for variable thin lines, this field specifies the height to adjust terrain to (instead of adjusting it up or down by this amount) when triggered.
  - For all other variable arcs, this the the amount to adjust the terrain by when the trigger activates.
- Use Trigger: Toggles the arc between being a static zshape and a variable zshape. Variable z shapes use triggers to change terrain when a specific event occurs. All variable Z Shapes have the Duration option to span the change over a certain period of time. Other options will change with the Trigger Type.
  - Trigger Type: The type of event that will trigger the Z modification.
    - Specified time: Triggers at a specific time during the simulation, measured in hours.
    - Water level at point: Uses a Trigger Point to measure water level, and triggers when the water level reaches or exceeds the value in the Water level field.
    - Water level difference: Uses two trigger points. Triggers when the difference between water levels measured at both points exceeds the value in Water level difference. A difference in either direction is treated the same in TUFLOW, and there is no need for negative numbers.
    - Thin line water level: Similar to Water level at point, but for thin lines. Does not use a trigger point, instead measuring the water levels on either side of the thin line.
    - Thin line water level difference: Triggers when the difference between water levels on either side of the thin line reaches or exceeds the value in Water level difference.

- Point1 and Point2: Both point selection boxes list the trigger points available in the coverage. These fields are only enabled for trigger types that use them.
- Specified time, Water level, Water level difference: These fields become available based on the trigger type selected.
- Duration: If set to 0, changes are instantaneous once triggered. Otherwise the change will be interpolated over this many hours.
- Restore Z Shape: This option can be used to have TUFLOW restore the original elevations at a specified time.
- Repeat restoration: The z-shape will be restored indefinitely.
- Restore interval: The time (hrs) between when the variable z-shape is finished and when to start restoring the points back to their original values.
- Restore period: The time (hrs) needed to restore the points back to their original values.

Trigger Points

Trigger points are used by the Water level at point and Water level difference trigger types. Simply create a feature point and enter its attributes dialog to create a trigger point.



Trigger Point Attributes Dialog

Trigger Point Dialog

The trigger point dialog is used to turn a feature point into a ZShape trigger point. Enabling the Specify Trigger checkbox and giving the point a name will cause it to appear in the Point1 and Point2 selection boxes in the arc attributes dialog for use as a trigger point. If you rename a trigger point that is being used by triggers, or delete it by unchecking Specify Trigger, SMS will notify you that there are one or more arcs/polygons being updated with the new change.

TUFLOW 2D Z Lines/Polygons (Simple) Coverage

These coverages are used as geometry modifications and force grid elevation values using arcs or polygons. This can be useful to ensure ridge or gully lines are represented in the model, simulate a proposed levy, or to simulate a proposed fill/excavation scenario. It is possible to only have the elevations assigned from a feature object if the feature elevation is higher or lower (user specified) than the existing elevations in the grid. This is controlled in the 2D Z Lines/Polygons (Simple) Coverage Properties dialog (right click on the coverage and choose Properties). The options specified in the coverage properties dialog apply to any geometry modification arcs and polygons within the coverage. If you want to use multiple settings within a simulation, you must have multiple 2D z line/polygon (simple) coverages.

Z Values

The z values determine how the elevations in the coverage are used to modify the existing model elevations. The options are:

1. All Zpts (default) – The z values from all of the arcs/polygons are used.
2. Min – The z values from the arcs/polygons will only overwrite the existing elevations if the elevations are lower.
3. Max – The z values from the arcs/polygons will only overwrite the existing elevations if the elevations are higher.
4. Add – The z value becomes the existing elevation plus the z value from the arc/polygon (which may be negative).

Thick Arcs

If this option is not selected, the z values of the arc will be applied to the nearest elevation locations in each cell that the arc passes through. Otherwise, the arc elevations will be applied to every elevation location in each cell that the arc passes through.

Make sure the coverage is in the Geometry Component(s) you wish to apply to the modification to.

Points

Points are ignored in the geometry modification coverage.

Arcs

Only the elevations at the nodes (endpoints) are used to set the grid elevations. Vertex (intermediate) elevations are ignored. Cell elevations along the arc will be interpolated linearly based on distance from the endpoint elevation. If you would like to specify elevations at vertices on an arc, convert the arc vertices to nodes to create multiple arcs joined end to end.

Polygons

To set the elevation for polygons double click on the polygon and enter the elevation in the dialog. This elevation will be applied to all cells within the polygon.

TUFLOW Boundary Conditions Coverage

Boundary conditions are defined in TUFLOW with points, arcs, or polygons in a BC coverage. BC coverages can also be used to specify cell code (active/inactive) areas of the 2D model domain. The kinds of boundary conditions available depend upon whether it is applied to a point, arc or polygon.

Boundary conditions defined at points are 1D boundary conditions and must be placed at the same location as a 1D boundary node. (See Snapping Feature Objects) The boundary condition information is specified in the BC Node Attributes Dialog (see BC Node Attributes Dialog).

Arcs can be used to define 2D Boundary conditions applied to the 2D domain. The BC attributes are specified in the BC Arc Attributes Dialog (see BC Arc Attributes Dialog).

Polygons can be used to define rainfall applied to 2D domains or to specify active/inactive information for 2D domains. The *BC Polygon Attributes* dialog specifies the type of information stored with the polygons (see BC Polygon Attributes Dialog).

Clip Region Read TIN Z pts Coverage

Using a coverage to specify a clip region, makes it so only elevations within coverage polygons are changed by a TIN dragged under a 2D geometry component. This is particularly useful for clipping out a TIN due to unwanted or irregular triangulation around the periphery, especially for secondary TINs of proposed developments lying within the primary TIN.

To use this coverage, create a coverage with one or more polygons. Only areas within polygons and within the TIN boundaries will have elevations assigned. You must have a scatter set in the 2D Geometry Components. Then drag and drop this coverage under the scatter tree item, under the 2D Geometry Components.

TUFLOW Grid Extents Coverage

A TUFLOW Grid Extents coverage is used to create TUFLOW grids. TUFLOW grids are created by creating and positioning a Cartesian Grid Frame and then right clicking on the coverage and choosing Map→2D Grid.

TUFLOW Water Level Lines Coverage

Water level line coverages define the locations where 1D solutions will be written as 2D output. TUFLOW creates water level points along the water level lines. The water level lines in conjunction with the water level points guide TUFLOW on how to represent the 1D flow in the 2D domain.

Arcs are the only objects used in the water level lines coverage. The only attribute in the water level lines coverage is the minimum distance between water level points. This is used by TUFLOW to generate the water level points along the arcs.

For more information about how TUFLOW uses water level lines to generate 2D flows see TUFLOW Water Level Points Coverage.

TUFLOW Water Level Points Coverage

Water level points are used in conjunction with water level lines to guide TUFLOW on creating 2D output for 1D networks (see TUFLOW Water Level Lines Coverage).

Water level points store a material value. This material value along with the z location of each point is used by TUFLOW to better approximate 2D flow along a water level line. TUFLOW performs a parallel channel analysis so that flow rates reported in the 2D output vary based upon the depth and roughness of a channel. Areas that are more rough and shallow than other areas will have a smaller flow rate than other areas along the same water level line.

The material value for each point can be mapped from an area property coverage by right clicking on the coverage tree item and selecting *Materials for Area Coverage*. By default the elevation of each point is based upon the neighboring cross-section information.

Creating the Water Level Points Coverage

The initial water level points coverage data is created by TUFLOW during a run. Once a run has been completed, the data can be read from the check file that starts with the simulation name and ends with _WLLp_check.mif (see TUFLOW check files). Open this file from SMS and select TUFLOW WLL Points from the combo-box that comes up. This coverage now can be included in simulations to provide additional guidance to TUFLOW in distributing 1D flows.

3.7.b. Interface Components

Interface Components

The map module interface consists of the display options, menus, right click menus, and tools associated with the map module.

- Menus
- Tools
- Project Explorer Items

Related Topics

- Display Options
- SMS Menus
- Dynamic Tools

Map Module Display Options

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The properties of all mapdata that SMS displays on the screen can be controlled through the map tab of the Display Options Dialog. This dialog is opened by right clicking on the Map Data entry in the Project Explorer and selecting the Display Options command. (It can also be accessed from the from the Display menu or the display options Macro.)

The exact layout of the Display Options dialog for feature objects depends on the active Coverage type. Some options are available on all coverages, while other options are only available on certain coverage types. The following options are available in the Display Options dialog. The entities associated with the map module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available map display options include the following:

Feature Objects

- Nodes
- Node IDs
- Refine Points
- Nodal Boundary Conditions
- Arcs
- Arc IDs
- Vertices
- Arc Boundary Conditions
- Polygon Fill
- Fill Colors
- Polygon IDs
- Legend
- Display of inactive Coverages

Drawing Objects

Each drawing object stores its own attributes specified by selecting the individual object. This dialog allows the user to turn on/off all drawing objects of a specific type.

- Text
- Lines
- Rectangles
- Ovals

Arcs

Display the feature arcs.

- ID – Display the ID of the feature arcs.
- Vertices – Display the vertices along the arcs.
- Arc types – Arc attributes may be displayed depending on the coverage.
- BC Display Opts – Open a dialog with display options for the boundary condition specific to the active coverage.
- CL Flow Direction Arrow – Display an arrowhead showing the direction of center line arcs.

Polygons (fill)

Color fill the feature polygons. The display of the boundary of the polygons is controlled by displaying feature arcs.

- ID – Display the ID of the feature arcs.
- Polygon types – Polygon attributes may be displayed depending on the coverage.
- Show Materials / Mesh Type – Fill the polygons to display the materials assigned to the polygons or the mesh generation type assigned to each polygon.
- Material Opts – Set the display options of the materials.

Grid Frame

Set the display of the grid frame (Cartesian grid coverages only).

Inactive coverage color

Set the color for the display of all inactive coverages.

Legend

Turn the legend on or off for feature objects. The Options button opens a dialog that controls the title, font, location, and size of the legend. The Active button in this dialog signifies to show only the active coverage in the legend and the All button signifies to show all coverages in the legend.

Background Color

Set the background color of the Graphics Window.

Observation Coverage Only

Calibration Target. Turn on/off calibration targets drawn next to observation points. The Interval and Two std. dev. options tell SMS to set the size of the calibration targets based on the interval or the standard deviation assigned to each point. The Scale tells SMS to scale the targets larger (>1.0) or smaller (<1.0) than the default size. **Computed.**

Tells SMS to Use the active data set or to Use the selected data set for displaying the calibration targets.

Related Topics

- Map Module
- Display Options

Map Module Menus

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The following menus are available in the Map Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Feature Objects Menu

Map Feature Objects Menu

The Feature Objects Menu includes the following commands:

General Commands

- Delete – This command deletes all the feature objects in an SMS session and creates a new blank coverage (since SMS requires an active coverage at all times). SMS will ask for confirmation of this action.
- Attributes
- Create Arc Group – This command will create a new entity from a group of selected contiguous (end to end) arcs. If the selected arcs are not connected end to end, SMS will give an error message. These arc groups can be used in some models to assign boundary conditions.
- **Build Polygons** – While most feature objects can be constructed with tools in the Tool Palette, polygons are constructed with the Build Polygons command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. After forming loops with arcs, choose *Feature Objects | Build Polygons* from the menu. The build polygons command will form polygons from all closed loops in the coverage.
- Clean
- **Vertices ↔ Nodes** – In some cases, it is necessary to split an arc into two arcs. This can be accomplished using the Vertex ↔ Node command. Before selecting this command, a vertex on the arc at the location where the arc is to be split should be selected. The selected vertex is converted to a node and the arc is split in two. The Vertex ↔ Node command can also be used to combine two adjacent arcs into a single arc. This is accomplished by converting the node joining the two arcs into a vertex. Two arcs can only be merged if no other arcs are connected to the node separating the arcs. Otherwise, the node must be preserved to define the junction between the branching arcs.
- Reverse Arc Direction
- Redistribute Vertices
- Transform Feature Objects
- Select/Delete Data...
- **Map → Scatter** – The Map → Scatter allows scatter sets to be interpolated from map data. Scatter points can be created from a specified source. Scatter points data can be extrapolated from feature points and vertices on arc or on feature points only on arc, feature points not on arcs or from feature polygon meshing options. The elevation source can be obtained from arc elevation, arc node and vertex elevations or from arc spacing.
- Map → 2D Mesh
- Map → 2D Grid

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

Stamping

- Stamp Features

Model Coverage Types

ADCIRC

- Model Control
- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens. For more information, see Create Contour Arcs. This command is available if the current coverage type is SHOALS, ADCIRC, or CGWAVE.
- Define Domain

BOUSS-2D

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.
- Extract Elevations – This option is found in the Feature Objects menu, in the Map module, when SHOALS is the active coverage type. The elevation of each node and vertex along every profile arc is interpolated from the active scatter set.

CGWAVE

- Model Control
- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.
- Define Domain

CMS-Flow

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.

CMS-Wave

- Create Coastline – When this command is invoked, the Create Contour Arcs dialog opens.

GenCade

- New Grid Frame

Related Topics

- Map Module

Map Module Tools

Map Module

Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

The following tools are contained in the dynamic portion of the Tool Palette when the Map Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the map tool palette.

Tool	Tool Name	Description
	Select Feature Point or Node	The Select Feature Point or Node tool is used to select stand alone feature points or the ends of arcs. A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature points are locked so they are not accidentally dragged. When a single point is selected, its location is shown in the Edit Window. The coordinates can be changed by typing in the edit field. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File | Info Options* command in the File Menu. Selected points can be deleted by selecting the *Edit | Delete* menu command on the Edit Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Arcs attached to the deleted points are deleted. This tool is available when one or more feature points exist.
	Create Feature Point	The Create Feature Point tool is used to place new feature point in the current coverage. A single point is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window. This tool is always available, however, creating a feature point is only allowed while in plan view.
	Select Feature Vertex	The Select Feature vertex tool is used to select one or more vertices on an arc. These vertices define the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.
	Create Feature Vertex	The Create Feature vertex tool is used to create a new vertex on the interior of an arc. The vertex is created at the current arc location, but can be selected and moved to change the shape of the arc. The vertex may have a "z" elevation specified, but no other attributes are associated with the feature vertices.

	Select Feature Arc	The Select Feature Arc tool is used to select one or more existing feature arcs. This is typically done to assign attributes to an arc or delete the arc. A single arc is selected by left-clicking directly on it. Double clicking on the arc will bring up the arc attributes dialog or that arc. Multiple arc can be selected at once by dragging a box. Additional arcs can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new arcs without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected arc can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Feature arcs may have elevations associated with the arc as an entity. This is independent of the node and vertex elevations. When a single arc is selected, its elevation is shown in the Edit Window. The coordinates can be changed by typing in the edit field. Multiple feature arcs may also be selected to create a feature arc group to associate attributes with a string of arcs rather than a single arc. These arcs must connect end to end. The arc group is created from the Feature Objects menu command. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File | Info Options* command in the File Menu. Selected arcs can be deleted by selecting the *Edit | Delete* menu command on the Edit Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Nodes attached only to the deleted arcs are deleted. This tool is available when one or more feature arcs exist.
	Create Feature Arc	The Create Feature Arc tool is used to create a new feature arc.
	Select Feature Arc Group	The Select Feature Arc Group tool is used to assign attributes to a string of arcs. The group must be created before it can be selected as a group. The attributes of the group then operate just as if the group was a single arc.
	Select Feature Polygon	The Select Feature Polygon tool is used to select polygons to assign attributes to them. The attributes associated with the polygon depends on the coverage type. Double clicking on the polygon will invoke the polygon attributes dialog for the polygon. Multiple polygons may be selected just as with arcs and nodes. The polygons must be created using the Feature Objects menu command.
	Select 1D Grid Frame	The Select 1D Grid Frame tool only appears when the coverage is associated with a 1D grid model (Cascade or GENESIS).
	Create 2D Grid Frame	The Create 2D Grid Frame tool is used to create a new grid frame for the creation of Cartesiona grids. The grid frame is defined by clicking three times in the graphics window. The first click defines the origin. The second click defines the I axis of the grid frame (both extents and direction). The third click defines the extents of the J axis. The direction is set to be perpendicular to the I axis.
	Select 2D Grid Frame	The Slect 2D Grid Frame tool allows the user to select a grid frame and alter its position, orientation and size.

General Tool Right Click Menus

The following is a list of options that appear in every tool's right click menu:

- **Clear Selection –** The Clear Selection command undoes the selection of the object that was clicked on.
- **Invert Selection –** The Invert Selection command selects every object of the same type selected, and undoes the selection of the object that was originally selected.
- **Zoom to Selection –** The Zoom to Selection command zooms to a closer view of the object selected.

Tool Specific Right Click Menus

Each tool in the Map module has its own right click menu. When the object is selected, one can right click and a menu will appear. The following is a table showing all of the different tools of the map module with their respective right click menus.

Tool	Tool Name	Right Click Menu
	Select Feature Point or Node	• Convert to Vertex/Vertices – Converts node(s) to Vertex/Vertices • Delete – Deletes node(s) • Transform – Moves the node(s) either by scaling, translation, or rotation. User specified.
	Select Feature Vertex	• Convert to Node(s) – Converts Vertex/Vertices to node(s). • Delete – Deletes vertex/Vertices • Transform – Moves the vertex/vertices either by scaling, translation, or rotation. User specified.
	Select Feature Arc	• Create Arc Group – Creates a group out of two or more arcs selected together. • Delete – Deletes arc(s) selected. • Split Arc(s) – Sub divides arc(s) at the vertices. • Offset Arc(s) – • Align Arc(s) with Contour – • Redistribute Vertices – User specified distribution of vertices. Vertices can be evenly distributed based on spacing or number of vertices desired. • Reverse Arc Direction – Reverses direction of arc. • Smooth Arc(s) – Smooths arc (sets the vertices of the arc more in line with each other) by setting a self weight, and number of neighbors. • Transform – Moves the arc either by scaling, translation, or rotation. User specified. • Attributes – Type specific. Many types do not have any attributes in their arcs. • Select Connected Arcs Turning Left – Selects the arc to the left of the originally selected arc.
	Select Feature Arc Group	• Delete – Deletes Feature Arc Group(s).
	Select Feature Polygon	• Delete – Deletes Feature Polygon (not the arcs that make up the polygon). • Attributes – Model specific.
	Select 1D Grid Frame	
	Select 2D Grid Frame	Type Specific • Properties – Shows the properties of the 2D grid frame. Model specific.

Related Topics

- Map Module

Project Explorer Items

In the Project Explorer, the Map data folder houses all of the coverages that are controlled by the Map module. The Map data folder can hold as many coverages as the user desires, and can also generate sub folders. Coverages are considered 'active' when clicked on in the Project Explorer, and the name of the coverage becomes bold while the coverage icon becomes green. When a new project is created in SMS, a coverage will be automatically created. This coverage is named 'default coverage', and it will be set to a default type which can be specified in 'Preferences'.

Map Module Right Click Menus

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a Map Module item.

Map Module Root Folder Right Click Menus

Right clicking on the Map module root folder in the project explorer invokes an options menu with the following options:

- **New coverage –** Opens the New Coverage Dialog.
- **New Folder –** Creates a new folder under the Map module root folder.
- **Clear Coverages –** Deletes all coverages.
- **Display Options –** Opens the Display Options Dialog

Coverage Item Right Click Menus

Right clicking on a Map item in the Project Explorer invokes an options menu with the following module specific options:

- **Duplicate –** Adds another coverage exactly identical to the existing coverage clicked on.
- **Rename –** Allows the user to specify a new name for the coverage.
- **Convert –** Converts coverage to a 2D grid, mesh or scatter object.
- **Projection –** Allows the user to set the projection of the coverage.
- **Reprojection –** Allows the user to reproject the projection of the coverage.
- **Metadata –** Allows the user to annotate the coverage.
- **Zoom to Coverage –** Zooms to area where coverage is within the graphic window.
- **Type –** Change the coverage type.

New Folder Right Click Menus

Right clicking on a new folder item in the Project Explorer invokes an options menu with the following options:

- **New Folder –** Creates a new sub folder under the new folder.
- **Delete –** Deletes the new folder.
- **Rename –** Allows the user to rename the new folder.

Right click options for the coverage may also include options applicable only to the specific coverage type.

3.7.c. Functionalities

Feature Objects Types

Map Module

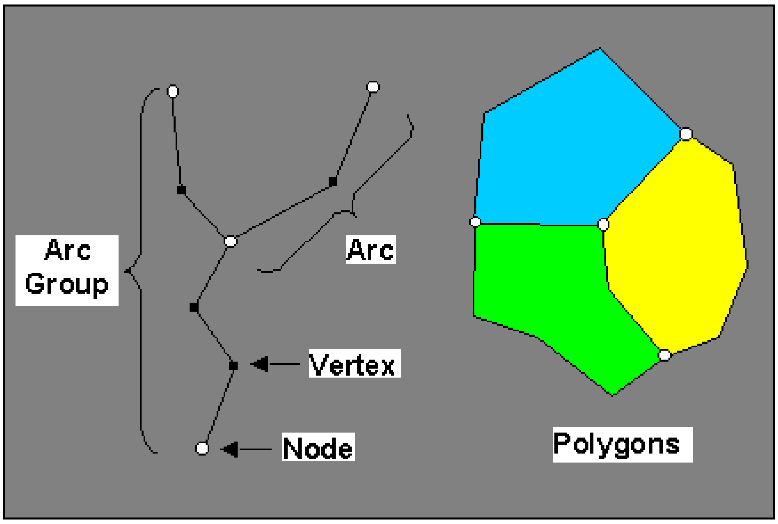
Map
Feature Objects
Coverages
More
Map Display Options
Map Module Tools
Map Module Menus

Feature objects in SMS have been patterned after Geographic Information Systems (GIS) objects and include points, nodes, arcs, and polygons. Feature objects can be grouped together into coverages. Each coverage defines a particular set of attributes that store information for the objects. Since feature objects are patterned after GIS objects, it is possible to import and export feature objects to files readable by GIS programs such as Arc/Info and ArcView.

The primary use of feature objects is generate high level conceptual representations of a site. The area included by the polygons defines the domain of the mesh, grid, or limit the extents of cross sections. Each polygon represents a material zone or element type. Special points can be identified in the interior of the domain as areas of particular interest. Boundary parameters such as flow and head values can also be assigned to points or arcs. Depending on the numerical model to be used, SMS either passes this conceptual representation to the model, or constructs finite element meshes, finite difference grids, or one-dimensional cross sections that a numerical model will use. Thus, the user can focus on a simplified, high level representation of the model and little or not tedious cell-by-cell editing is required. The conceptual model approach can be used to build models for any of the numeric models supported by the SMS interface.

Feature Object Types

The definition of feature objects in SMS follows that used by typical GIS software that supports vector data. The basic object types are points, nodes, vertices, arcs, and polygons. The relationship between these objects is illustrated in the figure below.



Feature Objects

Points

Points are XY locations that are not attached to an arc. Points have unique ids and can be assigned attributes such as a source or sink. Points are often used to refine a mesh in an area of interest. Points are also used when importing a set of XY locations for the purpose of creating arcs or polygons.

Arcs

Arcs are sequences of line segments or edges, which are grouped together as a single "polyline" entity. Arcs have unique ids and can be assigned attributes such as specified head. Arcs are grouped together to form polygons or are used independently to represent geometrical features such as ridges or channels. The two end points of an arc are called "nodes" and the intermediate points are called "vertices".

The vertices in an arc define the shape. As more vertices are added, the shape can be more complex. An arc is split into to arcs by selecting a vertex in the arc and converting that vertex to a node. Two adjacent arcs are merged into a single arc by selecting the node that joins them, and converting it to a vertex. Several tools or utilities are provided for working with arcs. These can be accessed while the select arc tool is active by right clicking in the graphics window. Some of the tools also require that at least one arc be selected. The tools include:

- Delete the selected (or all) arcs.
- Filter arc(s)
- Split arc(s)
- Offset arc(s)
- Redistribute vertices
- Reverse arc direction
- Smooth arc(s)
- Transform
- The standard selection utilities (clear, invert, zoom to selection)
- Select connected arcs

Nodes

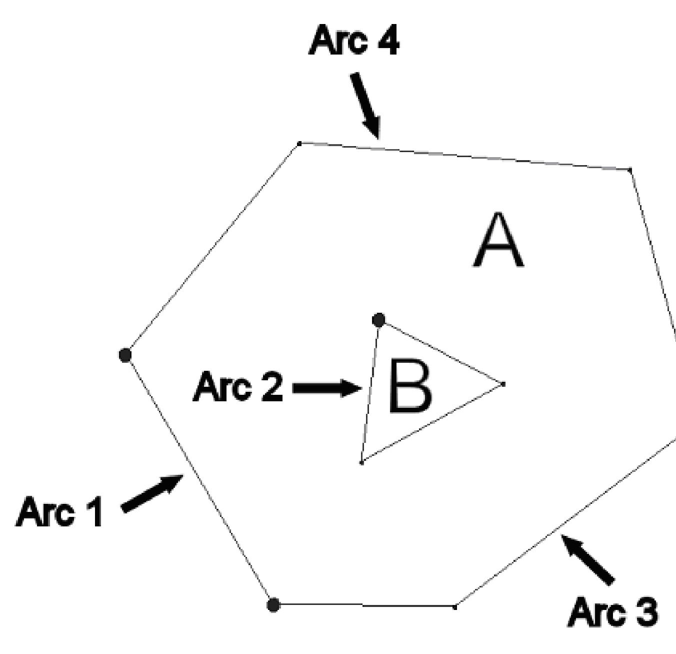
Nodes define the beginning and ending XY locations of an arc. Nodes have unique ids and can be assigned attributes.

Vertices

Vertices are XY locations along arcs in between the beginning and ending nodes. They are used solely to define the geometry of the arcs. Vertices do not have ids or attributes.

Polygons

Polygons are a group of connected arcs that form a closed loop. A polygon consists of one or more arcs. If two polygons are adjacent, the arc(s) forming the boundary between the polygons is shared (not duplicated). Polygons may not overlap. However, a polygon can have a hole(s) defined by having a set of closed arcs defining interior polygon(s). An example of a hole is shown in the figure below. In this case, four arcs define two polygons. Polygon A is made up of arcs 1, 2, 3 and 4, whereas polygon B is defined by a single arc (arc 2). For polygon A, arcs 1, 3, and 4 define the exterior boundary whereas arc 2 defines a hole.



Polygon with Hole

Polygons have unique ids and can be assigned attributes. Polygons are used to represent material zones such as main channel, overbank flood plain, lakes, etc.

Coverages

Feature objects are grouped together into coverages. Each coverage represents a particular set of data. For example, one coverage can be used to define recharge zones, and another coverage can be used to define zones of hydraulic conductivity.

Conceptual Models

Coverages are grouped into conceptual models. Conceptual models may consist of multiple coverages. In simple cases like TABS (RMA2), an "RMA2" meshing coverage may be combined with an "Area property" coverage to define material zones. When converting an RMA2 coverage to a mesh, SMS allows the user to specify this option. For more complex conceptual models, such as those used for TUFLOW, a simulation entry in the tree includes links to the component coverages. A TUFLOW simulation may have coverages for HX links, cross section, boundary conditions and levies.

Related Links

- Converting Feature Objects
- Feature Objects Menu
- Shapefiles

Attributes in Feature Objects Menu

Feature object attributes are dependent on the coverage type. If a feature object has attributes, the attributes are edited by selecting the feature object and then selecting the menu command Feature Objects | Attributes or Attributes from the right mouse menu.

Feature Point Attributes

Generic Coverage Types

- Observation
- Stamping

Model Coverage Types

- ADCIRC
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic 2D Mesh
- PTM
- TABS
- TUFLOW
  - 2D Flow Constriction Shape
  - BC
  - Network
  - Water Level Points
  - Z Shape

Feature Arc Attributes

Generic Coverage Types

- Observation
- Stamping

Model Coverage Types

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- GenCade
- Generic 2D Mesh
- PTM
- STWAVE
- TABS
- TUFLOW
    - 1D/2D Connections
    - 2D Flow Constriction Shape
    - 2D Z Lines/Polygons (Simple)
    - 2D/2D Linkage
    - BC
    - Cross Section
    - Grid Extents
    - Network
    - Water Level Lines
    - Z Shape

Polygon Attributes

Generic Coverage Types

- Area Property

Model Coverage Types

- ADCIRC
- BOUSS-2D
- CGWAVE
- CMS-Flow
- CMS-Wave
- FESWMS
- Generic 2D Mesh
- PTM
- STWAVE
- TABS

- TUFLOW
  - 2D Flow Constriction
  - 2D Z Lines/Polygons (Simple)
  - 2D Spatial Atts
  - 2D/2D Linkage
  - BC
  - Grid Extents

Related Topics

- Feature Objects Menu

Find (By ID)

Find

The 'Find' feature object command allows the user to find a feature object node, arc or polygon by its specified ID number.

Export (Convert to CAD)

For more information, go to CAD data.

Selection

Select By

The *Select By* option allows the user to select an object by its Material Type, Dataset Value, Area, Length, or Ambiguous Gradient.

Select Connected Arcs

As conceptual models become more complex, and include many arcs (possibly hundreds or thousands), detailed connectivity may not be visibly obvious. This means that you may create or import arcs defining model extents or other features, that appear to be connected, but in actuality are not.

The *Select Connected Arcs Turning Left* utility allows a user to easily determine if a conceptual model has gaps in connectivity. The utility is used by right-clicking on a feature arc and selecting a **"Select Connected Arcs Turning Left"** command. There are two options for selecting the arcs, Forward and Backward. The Forward option follows the direction of the arc and the Backward option follows the opposite direction. This utility selects a string of connected arcs. If more than two arcs connect at a node, the utility selects the left-most turn. In a completely defined polygon, this will select all the arcs in the polygon and traverse right back to the starting arc. If the select polygon tool fails to operate as expected, this utility may identify the gaps in connectivity causing the polygons to not be defined correctly by the build polygon command.

Select Connected Arcs Turning Left

This feature allows the user to select specific arcs attached to an arc already selected determined by the direction of the arc. The user can specify if they want to select the arc turning left from the perspective of going forward, or the arc turning left from the perspective of going backward. Both are opposing directions.

3.7.c.1. Feature Object Creation

Digitize

When you digitize in the map module, elevations are assigned as with other digitization in SMS. That means that when you create a node, point, or vertex, it is assigned the default elevation value for digitization. The default elevation is initialized to 0.0. The default changes any time you specify a Z-value. Therefore, if you create a map point or node, and specify an elevation for that selected point, the value you specify is now the default value for newly digitized points, nodes and vertices. (Note: when you create mesh nodes, there is an option to ask for an elevation each time you create a node, but this option is not available for scatter vertices or map module objects.)

If you would like to digitize aspects of an image (*.tif, *.jpg, ...), simply load the image file into SMS and select the desired tool from the Map module tools ('Create Feature Arc', 'Create Feature Point', etc.) and click over the part of the image that you would like to digitize.

Import Options

Map data can be imported into SMS by various means. For more information, go to Import Wizard.

Import From CAD/GIS

For more information, go to CAD data.

Convert From Mesh/Scatter

Other modules in SMS allow the user to convert data to map data. For more information, go to Mesh to Map and Scatter to Map.

Build Polygons

While most feature objects can be constructed with tools in the Tool Palette, polygons are constructed with the Build Polygons feature object command. Since polygons are defined by arcs, the first step in constructing a polygon is to create the arcs forming the boundary of the polygon. All closed loops will be formed into polygons. Before defining material zones or creating meshes from a coverage, the Build Polygons command must be used first.

Polygon Attributes

The *Polygon Attributes* dialog is used to set the attributes for feature polygons. Attributes that can be specified for each polygon include:

- Polygon Type
  - Land
  - Ocean

For models that generate meshes, there are certain polygon attributes that need to be set before the mesh can be created. For more information, see 2D Mesh Polygon Properties.

3.7.c.2. Feature Object Modification

All

Tansform

The Transform command is used to move scatter points. The user is asked which will be transformed, the active set or all sets. In the dialog that appears, the transformation type can be chosen and then appropriate parameters can be entered.

- Data can be scaled, translated, rotated
- Depths/Elevations can be converted back and forth

For more information, go to Transform (Data Menu).

Clean

The Clean command is used to fix errors in feature object data (Feature Objects menu, Map module).

- **Snap nodes –** This command will merge any two nodes or vertices together if they are within the Tolerance of each other. The new node will be placed at the location of one of the old nodes or vertices (see Figure A).
- **Snap selected nodes –** This command merges two or more selected nodes or vertices. The user must click on one of the selected points which will be treated as the new location (see Figure A).
- **Intersect arcs –** This command places a node where two arcs intersect. All intersections are fixed with this command (see Figure B).

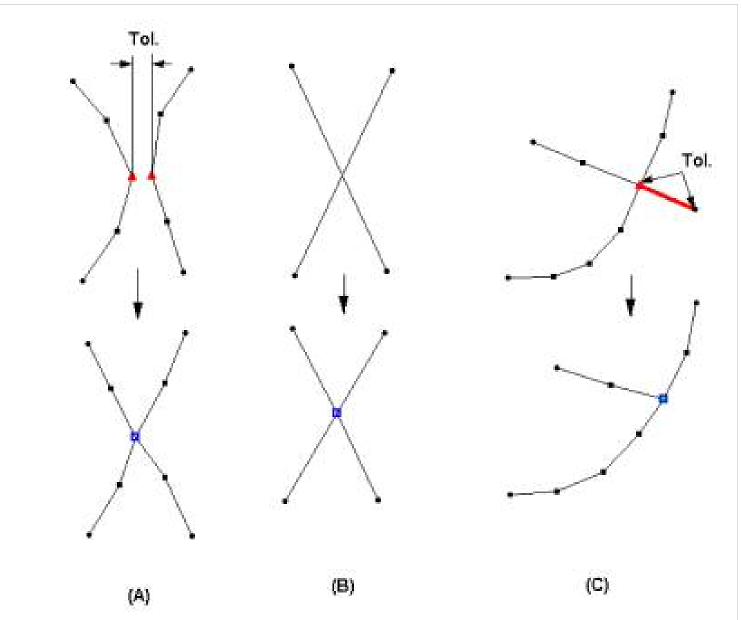


Figure C: (A) snapping (B) intersecting (C) dangling arcs

- **Intersect selected arcs –** This command again places a node where two arcs intersect, but only selected arcs are checked.
- **Remove dangling arcs –** The user specifies a Tolerance and all dangling arc segments (at least one end of the arc is not connected to another arc) are deleted if their length is less than the tolerance (see Figure C).

Delete

Delete all feature objects including all coverages and entities in the coverages. A new, empty coverage is created because there must always be a coverage in SMS. This does not delete Drawing Objects, DXF, or images.

Converting Coverages

Map coverages can be converted to other data types in SMS such as meshes, grids, scatter sets and cross sections. This can be accomplished by either right-clicking on a coverage in the project explorer and selecting a convert command or by selecting the following commands from the Feature Objects menu:

Extract Cross Section

The Extract Cross-sections command uses the cross section arcs and a digital terrain model (TINs are the only source that can currently be used) to extract the elevations at vertices of the feature arc cross-sections, or at the intersection points with the triangles.

Cross-sections for individual arcs may be extracted by selecting the arc(s) before choosing the Extract Cross-sections command. If not cross-sections are selected then the Use All Cross-sections option is used.

Point properties (thalweg, left bank, right bank) can be defined from a 1D-Hydraulic Centerline coverage, or by AutoMark. The AutoMark option will examine the elevations of the extracted cross sections and try to infer the thalweg (low point) and the left and right bank points (change of slope) automatically.

Line properties can be determined from an area property coverage by intersecting the cross-section arcs with the area property polygons and marking them in the cross section database.

Cross Section Database

When extracting the cross sections you will be prompted for the name of a cross-section database file. SMS stores all of the cross-section information in a text database file. The cross section database can also be edited independently using the Cross Section Editor tools. Extracting cross sections with feature arcs is the only way to generate cross-section information, they also can be imported from spreadsheet files (cut and paste), or entered manually.

Map → 2D Mesh

Once a set of feature objects has been created for a coverage (conceptual model) associated with a finite element based model such as RMA2, FESWMS, ADCIRC or CGWAVE , the Map → 2D Mesh command can be used to generate a 2D finite element mesh from the objects. The Map → 2D Mesh command creates a 2D Mesh on the interior of all of the polygons in the current coverage. The figure domain of a flood plain using the feature objects in the Map Module. The second figure shows a 2D Mesh created from the polygons.

Map → 2D Grid

The Map → 2D Grid command is used to create a 2D grid using the feature objects in a 2D Grid Coverage. When the Map → 2D Grid command is selected, the Create Grid dialog appears. A grid frame must have been defined. The size and location of the grid frame are used to initialize the fields in the Create Grid dialog. In most cases, these values will not need to be changed and the user can simply select the OK button to create the grid. If a grid frame has not been defined, the size and location of the grid are initialized so that the grid just surrounds the currently defined feature objects. If desired, the grid dimensions can be edited prior to selecting the OK button to create the grid.

Grid Frame Properties

The grid frame properties dialog allows the user to specify the attributes applied to the grid frame when performing a "Map → 2D Grid" operation. These properties are as follows:

- Origin
- Orientation
- Directional properties (u and v direction)
  - Define cell sizes – specified uniform cell sizes
    - Cell size – the cell size in the specified direction
    - Number of cells – number of cells in the specified direction
  - Use refine points – refine points will be used to generate the grid
    - Maximum cell size – the max size the should exists when growing
    - Maximum bias – the max growth ratio to be used when growing
    - Use inner growth – specifies whether the cell sizes should grow between two refine points
  - Grid size – the grid dimension in the specified direction

When the user specifies "Define cell sizes", there are a few options available. These options are:

1. Specify cell size – specify the cell size and the number of cells will be computed.
2. Specify number of cells – specify the number of cells and the cell size will be computed.

If the grid is to have square cells, the v direction cell size will always be linked to the u direction cell size.

Refine Points

Refine points for a Cartesian Grid allow a user to change the cell dimensions when generating the grid. They are not available for all models, since some Cartesian Grid models require uniform cell sizes. The user can specify whether to refine in the I and/or J direction and the base cell size for each direction.

When the refining is performed, the base size may be changed in order to fit the other restrictions applied to the refining process. If two refine points are too close to each other to allow the cell size to transition, one will be ignored when generating the grid. See Refine Point Dialog for more information.

Depth and Vector Options

In addition to the options specified on the grid frame, depth and vector interpolation options can be specified during the mapping process for some models. The depth mapping is required for all models, while the vector mapping is optional even for the models it can be performed on. Depth and vector datasets can be constant or interpolated from a scatter set.

Cells a user specified tolerance above the datum can be marked as land (inactive) cells. This option is on by default for BOUSS-2D, but defaults to off for other models.

When specifying a constant vector, the X and Y components are oriented based on global space, not grid space.

The name of the vector dataset can be specified, but the name of the depth dataset is always set to "Depth".

Map to 2D Scatter Points

The Map → 2D Scatter Points command creates a scatter point set from the points and nodes and vertices of the current coverage. The process is different for observation coverages and non-observation coverages.

- **Non-observation Coverages**

  With non-observation coverages, a single elevation dataset is created for the 2D scatter points representing the Z location of all the points, nodes and vertices.

- **Observation Coverages**

  With observation coverages, the Observation Points -> Scatter Points dialog appears. This dialog allows you to create a dataset for the 2D scatter points from one of the measurements associated with the observation points.

Measurement

A dataset is created for the 2D scatter points from the measurement selected in the dialog. The model associated with the selected measurement (if any) is shown, along with whether the measurement is steady state or transient.

Time Step Times

This section of the dialog is only available if the selected measurement is transient. It allows you to define the number of time steps, and the time step times to be created for the scatter point dataset.

- **Match all unique times**

  The Match all unique times option gets the set of unique times from the XY series of all the observation points. This is the union of all the times. If some XY series use dates/times and others don't, this option won't be available. Otherwise, the times in the spreadsheet will be displayed as either dates/times or relative times depending on the XY series. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the observation point XY series use dates/times or not. The Reference time section will be unavailable, but if the XY series use dates/times, the minimum time will be used as the reference time for the scatter point dataset.

- **Match time steps from model**

  The Match time steps from model option will only be available if the measurement is associated with a model, and the model is transient. If so, this will be the default choice and GMS will get the times to display in the spreadsheet from the stress period and time step info for the model. The spreadsheet will not be editable. The Use dates/times toggle will be unavailable but set according to whether the model uses dates/times or not. The Reference time section will be unavailable, but if the model uses dates/times, the model reference time will be used as the reference time.

- **Specify times**

  The spreadsheet of times will be editable with this option and you can copy and paste times from another program such as a spreasheet. Also, the Initialize Times button becomes available allowing you to bring up a dialog you can use to create times at a specified interval. If you select the Use dates/times toggle, the Reference time section will become available and the times in the spreadsheets will be displayed as dates/times.

Unstructured Grid Generation from a Conceptual Model

Traditionally, the most time consuming component of using a multi-dimensional hydrodynamic numerical model has been the generation of unstructured grids (also called meshes). This effort has given models based on Cartesian grids (structured grids) a decided simplifying advantage. Digitizing node points and connecting them into elements, while seemingly not a complicated process, becomes overwhelming when you consider the number of nodes and elements that compose a numeric simulation (thousands to even millions and the number is still growing).

The SMS interface includes the capability to define a 2D mesh using the feature objects in a 2D Mesh Coverage. When the Map → 2D Mesh command is selected, the 2D Mesh Options dialog opens. A meshing polygon must have been defined prior to issuing this command. The attributes of the meshing polygon(s) are used to generate the 2D mesh.

The meshing options are used with coverages that generate meshes for specific numeric engines. Some of the options may not be available for all coverage types since some models have specific requirements such as a limited number of supported element types.

Feature Polygon attributes

The process of generating a mesh involves filling the polygons in the coverage with elements. These elements can be triangular or quadrilateral depending on the numeric engine they will be used with. The user specifies how the polygons will be filled choosing from the following "Mesh Type" options:

- None – there will be no elements in this polygon. This will represent an island in the domain.
- Patch – the mesh is topologically a triangle or rectangle that will be filled with elements that conform to its sides.
- Paving – the mesh will be filled with elements by offsetting from the boundaries. The distribution of the vertices on the arcs comprising the polygon control the mesh density.
- Scalar Paving Density – the mesh will be filled with elements using the paving approach, but the distribution of vertices along the arcs, and throughout the interior, will be controlled by a scalar dataset specifying a target edge length.
- Existing Nodes – the polygon will be used as a stencil to keep the mesh nodes and elements already in that region.

In addition to specifying the method to fill the polygon, the user also specifies the source of bathymetric or topographic elevation data for the newly constructed mesh using the "Bathymetry Type" controls. The user can choose from:

- Constant – all newly created nodes will be assigned a single specified value.
- Scatter Set – all newly created nodes will be assigned a value based on interpolation from a scatter set or TIN.
- Raster – all newly created nodes will be assigned a value based on interpolation from a Raster object.
- Exiting Mesh – all newly created nodes will be assigned a value based on interpolation from the previously existing mesh.

The user also has control to specify the material type that will be assigned to newly created elements.

Feature Arc attributes

The feature arcs in the conceptual model serve three purposes.

1. They carry boundary condition attributes for the specific model or engine.
2. They control final mesh density for the paving or patch options.
3. They control detailed feature maintenance when they lie inside of paved polygons. An arc representing a thalweg (channel), crest, ridge, or shoulder will be incorporated into the meshing pattern to ensure that these features are maintained.

Feature Point attributes

Feature points can be included in polygons to carry a boundary condition such as a source or sink, or they can be used to control resolution in a specific area. The feature point can be assigned an attribute to be a refine point. In this case, the user specifies the size of element around that location. The mesh generation process will generate an element or cluster of elements at that location matching the specified size. These are then incorporated into the surrounding mesh using the advancing front paving method.

Feature points can also be assigned an attribute to control whether a mesh node will be incorporated into the mesh at the exact feature point location.

Nodes/Vertices

Switch Vertices and Nodes

In some cases, it is necessary to split an arc into two arcs. This can be accomplished using the Vertex ↔ Node command. Before selecting this command, a vertex on the arc at the location where the arc is to be split should be selected. The selected vertex is converted to a node and the arc is split in two.

The Node ↔ Vertex command can also be used to combine two adjacent arcs into a single arc. This is accomplished by converting the node joining the two arcs into a vertex. Two arcs can only be merged if no other arcs are connected to the node separating the arcs. Otherwise, the node must be preserved to define the junction between the branching arcs.

Select/Delete Data...

Select/Delete Data... (Feature Objects Menu) The Trim Data command (Feature Objects menu, Map module) is available when one or more polygons are selected. The user may select or delete data that is located inside or outside of selected polygons. Options are provided to select or delete mesh node, elements or duplicate nodes, scatter points or triangle, Cartesian grid cells or cell location that are partially (triangles/elements that cross a boundary) or completely inside/outside the polygons.

Function Type

- **Select –** Select objects
- **Delete –** Delete objects

Data Domain

- **Inside polygons(s) –** Trim data inside selected polygons.
- **Outside polygon(s) –** Trim data outside selected polygons.
- **Treat boundary as [Outside | Inside]** - Treat data that lies on polygon boundaries as if it were outside/inside of the polygon(s).

Select Data Type

Choose the data type to select or delete

- **Mesh**
  - **Nodes** (Select) – Select mesh nodes only.
  - **Nodes and Elements** (Delete) – Trim mesh nodes and elements.
  - **Elements –** Select or delete mesh elements only.
- **Scatter**
  - **Points** (Select) – Select scatter points only
  - **Points and Triangles** (Delete) – Trim scatter points and triangles.
  - **Triangles –** Trim triangles only.
  - **Scatter Sets to Trim –** Click on a scatter set in the window to toggle it on/off for trimming. Push the Active Set button to only select the active scatter set and the All Sets button to turn on/off all sets. Points and/or triangles are deleted only from the selected sets.
- **Cartesian grid cells**
- **Cartesian grid cell locations –** this can be useful in working with elevation values in [[TUFLOW grids.

Arcs

Reverse Arc Direction (Feature Objects Menu)

This command (Feature Objects menu, Map module) reverses the direction of all selected arcs. Arc orientation is only important for applications that use coastline or centerline arcs that have an arrow drawn on the arc. Each arc has a direction. One node is the "from" node, the other node is the "to" node. For most applications, the direction of the arc does not matter. However, when the arc is used to define a observation plots, and in various other situations the direction of the arc becomes significant. The Reverse Arc Direction command can be used to change the direction (upstream to downstream) for an arc.

Align Arc With Contour (Right-click Menu)

This command (arc right-click) can only be used when the project has scatter data. A dialog is brought up from which a scatter dataset, dataset value, and maximum distance are set. SMS will then move the vertices along the selected arc to locations where the scatter dataset value matches the dataset value specified in the dialog, as long as it is not moved by a distance greater than the maximum distance (also specified in the dialog).

Offset Arc (Right-click Menu)

The offset arc command (arc right-click) provides a mechanism to create additional arcs in a coverage by offsetting from the selected arc(s). Multiple offsets can be created in a single command. This command brings up a dialog that asks the user for a number of parameters including:

- The direction of the offset (to the left of the arc, to the right of the arc or both).
- The number of arcs to create in the specified direction.
- The maximum offset in the specified direction. If more than one offset is created on a side, the offset provided is for the last offset and the others are interpolated between the original arc and the specified offset. (If to arcs are created on the left side with an offset of 10.0, the first is offset by 5 and the second is offset by 10.)

This command uses the orientation of the selected arc as a basis meaning it knows one end of the arc as the "start" and the other as the "end". This convention is determined based on the creation of the arc. This direction is not always obvious and the user may need to investigate using trial and error which end is the start and which end is the end (as with bias in the redistribute command).

This command has several useful applications. These include:

- When defining a conceptual model, the base line data may include a feature such as a thalweg or a toe of a bank, or a center-line of a road or levee. The offset command can be used to approximate the opposing bank, a shoulder from a toe or similar, nearly parallel feature.
- When creating arcs that define a tropical storm path, variations of that path may be desired. The offset command creates arcs to represent these paths.

Redistribute Vertices (Feature Objects Menu)

The primary function of the vertices of an arc is to define the geometry of the arc. If the arcs are to be used for automatic mesh generation, the spacing of the vertices is important. The spacing of the vertices defines the density of the elements in the resulting mesh. Each edge defined by a pair of vertices becomes the edge of an element. The mesh gradation is controlled by defining closely spaced vertices in regions where the mesh is to be dense and widely spaced vertices in regions where the mesh is to be coarse.

When spacing vertices along arcs, the Redistribute command in the Feature Objects menu can be used to automatically create a new set of vertices along a selected set of arcs at either a higher or lower density. The desired arc(s) should be selected prior to selecting the Redistribute command.

The current status of the selected arc(s) is given at the top of the dialog. This includes the number of segments and spacing of those segments. When multiple arcs are selected, the current status is a combination of all selected arcs. However, the parameters set in this dialog apply to each arc individually. Therefore if multiple arcs were selected, each arc would reflect the options selected in this dialog. The following options are available for redistributing vertices:

Linear Interpolation

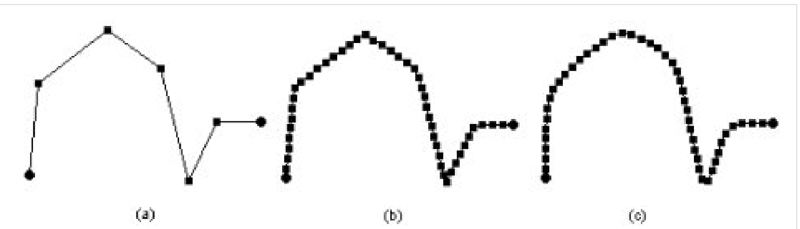
If the Linear interpolation options is specified, then either a number of intervals or a target spacing can be given to determine how points are redistributed along the selected arcs. In either case, the new vertices are positioned along a linear interpolation of the original arc. The arc may change shape due to the fact that original vertices are removed as the new vertices are created. This may round corners from the arc.

Spline Interpolation

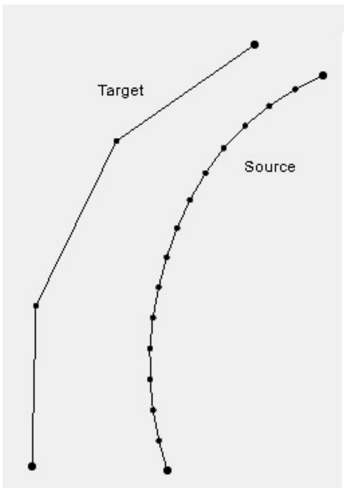
If the Spline interpolation option is specified, vertices are redistributed along a series of cubic splines defined by the original vertices of the selected arcs. The difference between the linear and spline interpolation methods is illustrated in the figure below.

Source Arc

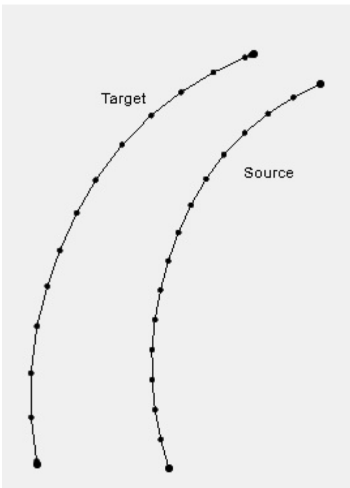
The Source arc option is only available when two arcs are selected. One arc is specified to be the source arc while the other is the target arc. SMS redistributes the vertices on the target arc to be as close as possible to the vertices on the source arc.



Redistributing Vertices: (a) Original Arc (b) Linear (c) Spline



Before Redistribution



After Redistribution

Split Feature Arcs Utility

Arcs are used to represent features in a conceptual model. A feature arc may represent a very small (short) individual feature, or it may represent a convoluted complex feature such as an entire shoreline. Two sample applications include arcs representing a shoreline, or arcs representing the thalweg of a river. There are advantages to having a single arc represent a long feature. These include the ability to select the entire feature easily, it is less cumbersome to manage the feature because there are less features, and you can ensure that there are no gaps in the feature because it is a single arc. However, there are also applications and purposes for splitting the arc into pieces. One such example is illustrated by the change of shape an arc undergoes when it is redistributed. The vertices move, but the feature nodes do not. In this situation, making key points nodes prevents the arcs shape from loosing its ability to represent these key locations. In the case of a thalweg, it could be advantageous to have a separate arc for each section of the river. Each arc would then span from one river station to another (perhaps every 100 yards or meters, or each river mile).

This utility is in the right click menu when you select one or more arcs. The command brings up the "Split Arcs Tool" dialog. In this dialog the user provides criterion for splitting the selected arc(s) into multiple arcs. If the user desires to process all the arcs in a coverage, the user can “select all” first.

There dialog includes three toggle boxes associated with criteria including:

- **Split long arcs** – if this is selected, the user specifies a length (labeled as ft or m) based on the current projection. Processed arcs will be split into arcs of this specified length starting at the first of the arc. The last arc in the group will generally not be the specified length.
- **Split sharp corners** – if this is selected, the user specifies an angle (labeled as degrees). This is the angle of deviation from one arc segment to the next. A straight line has an angle of deviation of 0.0. If the arc doubled back on itself completely, the angle of deviation is 180 degrees. This deviation is a magnitude. A left turn is processed identically to a right turn. Processed arcs will be split at vertices where the arc direction changes more than the specified angle.
- **Split long segments** – if this is selected, the user specifies a threshold length. If a segment is longer than this threshold, the vertices at either end will be converted to nodes making it a separate arc.

If multiple toggles are selected, the arc will first be split based on maximum arc length, the resulting arcs will be split based on angle, and finally the resulting arcs will be processed for long segments. This will retain the feature of creating feature nodes at the key length locations, and then add additional feature nodes at bends.

Select Connected Arcs

As conceptual models become more complex, and include many arcs (possibly hundreds or thousands), detailed connectivity may not be visibly obvious. This means that you may create or import arcs defining model extents or other features, that appear to be connected, but in actuality are not.

The *Select Connected Arcs Turning Left* utility allows a user to easily determine if a conceptual model has gaps in connectivity. The utility is used by right-clicking on a feature arc and selecting a **"Select Connected Arcs Turning Left"** command. There are two options for selecting the arcs, Forward and Backward. The Forward option follows the direction of the arc and the Backward option follows the opposite direction. This utility selects a string of connected arcs. If more than two arcs connect at a node, the utility selects the left-most turn. In a completely defined polygon, this will select all the arcs in the polygon and traverse right back to the starting arc. If the select polygon tool fails to operate as expected, this utility may identify the gaps in connectivity causing the polygons to not be defined correctly by the build polygon command.

Select Connected Arcs Turning Left

This feature allows the user to select specific arcs attached to an arc already selected determined by the direction of the arc. The user can specify if they want to select the arc turning left from the perspective of going forward, or the arc turning left from the perspective of going backward. Both are opposing directions.

Create Contour Arcs

The *Create Contour Arcs* dialog is used to create a feature arc along a specified elevation of the active scatter dataset or cartesian grid elevation. To access the dialog do the following:

- Right click on a Cartesian Grid or Scatter set in the Project Explorer.
- Select *Convert* then *2D Grid Contours→Map* for Cartesian Grid, otherwise select *Convert* then *Scatter Contours→Map*. This brings up the *Create Contour Arcs* dialog. By default the active coverage is selected. The user is prompted for the elevation and vertex spacing along contour.

Arc Attributes Dialog

The Arc Attributes dialog is used to set the attributes for feature arcs. Attributes that can be specified for each arc include:

- Arc Type
  - Generic
  - Percent preference coastline
  - Land preference coastline
  - Ocean preference coastline

Smooth Arc (Right-click Menu)

This command reduces the variability or roughness of an arc. When an arc is created by digitization, it includes the manual variations and noise from the digitization process. Similarly, when an arc is created to follow a contour line, it can have numerically created corners and bends. Smoothing an arc results in more gradual bends in the shape of the arc. Applying the smooth command repetitively will eventually result in a straight line.

An arc should not be smoothed, if the meanders and bends accurately depict a physical feature. However, smoothing may result in more gradual variations that can enhance numerical stability.

The dialog that appears when this command is invoked asks the user to specify the number of neighbors to be included in the smoothing window and a self weight. As the number of neighbors increases and the self weight decreases, the level of smoothing becomes more dramatic (moves to a straight line more quickly).

The algorithm computes a new position for each vertex in the arc from the existing vertex positions. The number of neighbors must be the same both before the vertex being smoothed and after. Since there are no vertices before the first node, and none after the last node, these two locations are not impacted by the smoothing process. The first vertex can be smoothed using the first node from before its position, and the first vertex after its position.

If the self weight is set to 1.0, no influence is assigned to the neighbors, so the arc does not smooth at all. If the self weight is set to 1/3, and the number of neighbors is set to 1, then each of the points has an even 1/3 weight in determining the new vertex location. If the self weight is set to 1/5, and the number of neighbors is set to 2, then five vertices impact the new location (2 vertices before the vertex in question and 2 vertices after) and the resulting point is the average of those five points (20% self weight, 80% neighbor weight with four neighbors).

Recompute All Stations (Feature Objects Menu)

This function can be used if an arc's "Computational Length" value on a river changes. It sets the Start and End station values for each of the arcs with the same river name based on the computational lengths of each arc. So, for example, if you had 3 arcs in a river, the most downstream arc being 30 feet, the next arc being 40 feet, and the next arc being 50 feet, the starting and ending stations would be as follows for each of the arcs (Arc 1 is downstream):

Arc	Start Station	End Station
1	0.0	30.0
2	30.0	70
3	70.0	120.0

These start and end stations are set when you assign a river name to an arc, so you should really never need the "recompute all stations" command unless you change the computational length, start station, or end station values for a centerline arc.

Feature Object Commands

General Feature Object Commands

Define Domain (Feature Objects Menu)

This option is found in the Feature Objects menu, in the Map module, when CGWAVE or ADCIRC is the active coverage type. A domain is the region to be filled by a finite element mesh. An ocean arc is created that connects to the coastline arc. There are several options for defining the domain:

1. User selects one or more coastline arcs. The ocean arc starts and ends at the extreme ends of the chain of arcs.
2. User selects two vertices or two nodes. The ocean arc starts and ends at the nodes.
3. User selects a single node or vertex along the coastline arc. The user defines the radius of the ocean arc and the ocean arc starts and ends where it intersects the coastline arc.
4. User selects a single disjoint point. If the point is inside a closed coastline arc (an island), the user specifies the radius of the ocean arc. The ocean arc is a circle that encloses the island.

After the user selects an arc(s) or point(s), the Domain Options dialog appears with the following options:

- **Rectangular –** If the user selects two points or an arc, the user only sets the Offshore Length. If the user specifies a single point, the user sets both Offshore and Along Shore Lengths.
- **Semi-circular –** If the user selects two points or an arc, the user does not specify the radius, the radius is the distance between the points. If the user specifies a single point, the user sets the Radius.
- **Circular –** This option only works if the user selects one point. The user then sets the Radius.

Right Click Menu

- **Clear Selection (Right Click Menu) –** The *Clear Selection* command undoes the selection of the object that was clicked on.
- **Invert Selection (Right Click Menu) –** The *Invert Selection* command selects every object of the same type selected, and undoes the selection of the object that was originally selected.
- **Zoom to Selection (Right Click Menu) –** The *Zoom to Selection* command zooms to a closer view of the object selected.

Coverage Type Specific Menus

Optional menu items appear according to the active coverage type.

Generic Coverage Types

- **Create Coastline**

  When this command is invoked, the Create Contour Arcs dialog opens. For more information, see Create Contour Arcs. This command is available if the current coverage type is ADCIRC, or CGWAVE.

- **Stamping**

  See the article Feature Stamping for more information.

Model Coverage Types

- **ADCIRC**
  - Model Control
  - Create Coastline – see above for more information.
  - Define Domain – see above for more information.
- **BOUSS-2D**
  - Create Coastline – see above for more information.
- **CGWAVE**
  - Model Control
  - Create Coastline – see above for more information.
  - Define Domain – see above for more information.
- **CMS-Flow**
  - Create Coastline – see above for more information.
- **CMS-Wave**
  - Create Coastline – see above for more information.
- **GenCade**
  - New Grid Frame – see the article Grid Frame Dialog for more information.

Related Topics

- Map Module

3.8. Mesh Module

Mesh Module

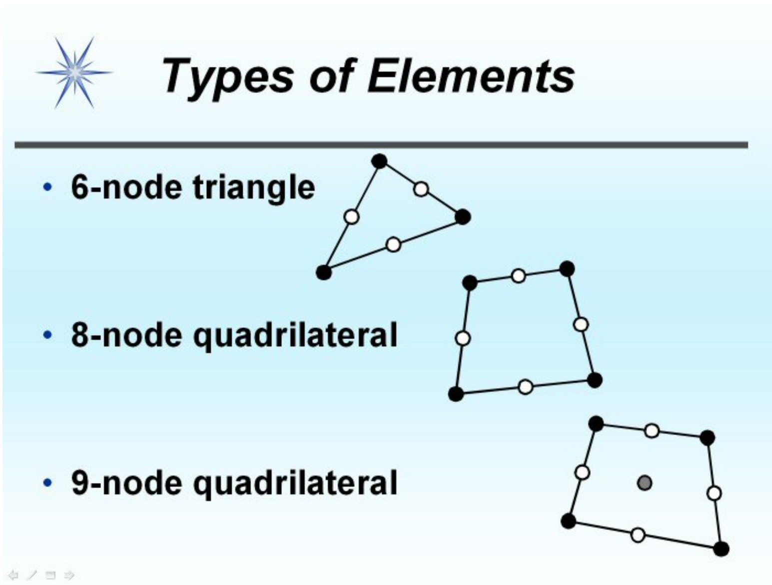
The 2D Mesh Module is used to manipulate 2D unstructured grids (referred to as a mesh inside of SMS). A mesh consists of nodes that are grouped together to form elements. These nodes and elements define the computational domain of the numerical model. In addition to nodes and elements, a mesh may assign additional information to the nodes and elements, such as material values assigned to elements and boundary conditions assigned to nodes. In general, this additional information is used as input data for the numerical model.

Nodes

Nodes are the basic building blocks of elements in a mesh. Nodes store elevation and other dataset values. Nodes can also be used for building nodestrings and assigning boundary conditions. The density of mesh nodes helps determine the quality of solution data and can be important to model stability. See 2D Mesh Nodes Menu for more information.

Elements

Elements are used to describe the area to be modeled. Elements are formed by joining nodes. The element types supported vary from model to model. Element types include:



Mesh Element Types

1D elements

- Three-node line

Triangular Elements

- Three-node linear triangule
- Six-node quadratic triangle

Quadratic (order of solution) Elements

- Eight-node “serendipity” quadrilateral
- Nine-node “Lagrangian” quadrilateral

Water surface and ground elevations are interpolated linearly within each element based on values at the corner nodes. Velocity is interpolated using a quadratic approximation based on values at all the nodes of the element. The quadrilateral elements use identical linear interpolation functions, but their quadratic functions differ because of the presence of an additional node at the center of the nine-node quadrilateral element.

Mesh Datasets

Datasets in a mesh store scalar or vector values at each node. Dataset Active/Inactive areas (generally indicating wet vs dry areas) can be defined on elements or be determined from specific NULL values (often -999) on the nodes themselves. Datasets are used for input data primarily for bathymetry values. Datasets are generally the primary output from a numeric model. Datasets are used with visualization options such as contours and vectors as well as data extraction tools like observation profile plots.

Node/Element properties

In addition to datasets, nodes and elements sometimes need to store additional information generally for model setup. The most common example of this is material types which are assigned to elements.

Nodestrings

A collection of nodes can be formed into a nodestring. Nodestrings are most commonly used to assign boundary conditions such as a flowrate or water-surface elevation. Nodestrings can also be used for mesh renumbering, forcing break lines, and boundary smoothing. Finally, a nodestring can store attributes pertinent to a location such as the total flow nodestring.

Mesh Generation

See Mesh Generation for more information.

Editing a Mesh

Whenever practical a mesh should be reconstructed from a conceptual model rather than edited in the mesh module. Often this isn't an option and a mesh must be edited by hand. The 2D Mesh Module Tools are used to create and edit meshes within the mesh module.

Mesh Visualization

After an analysis, output data at each node of the mesh can be used to generate linear or color filled contours as well as display vector arrows to visualize model solutions. Animations can be generated that shows changes through time for a time-varying solution. Meshes can also be used with flowtrace and multiple view animations.

Advantages of a Mesh

Meshes and other types of unstructured grids (such as TINs) have the advantage they can include a wide range of element sizes and transition gradually between them. Coastal models are often extreme examples of this sometimes with elements as small as a few meters and as large as many kilometers in the same domain. This allows a very large domain while keeping model computation time to a reasonable level.

Mesh Models

SMS has interfaces to several models that use meshes for computations and boundary conditions. These include:

- CGWAVE
- FESWMS-2DH
- ADCIRC
- TABS – (RMA2, RMA4)

In addition to the models with complete interfaces, the mesh module contains the Generic Model interface which allows users to customize SMS to generate data for a user defined model.

The finite difference model TUFLOW uses meshes for post-processing but not for building model domains.

Current Numerical Model

The mesh module is set up to be used with a single numerical model analysis engine at any given time. The current numerical model is changed using the Data Menu, Switch Current Model menu command. SMS shows only those tools in the tool palette and those menus in the Menu Bar which are relevant to the current numerical model. After a finite element mesh has been read, boundary conditions and material properties can be assigned using the commands in the menus associated with the current numerical model. The current model on startup can be changed in the Preferences Dialog.

Mesh Module Tools

See Mesh Module Tools for more information.

Mesh Module Menus

See 2D Mesh Module Menus for more information.

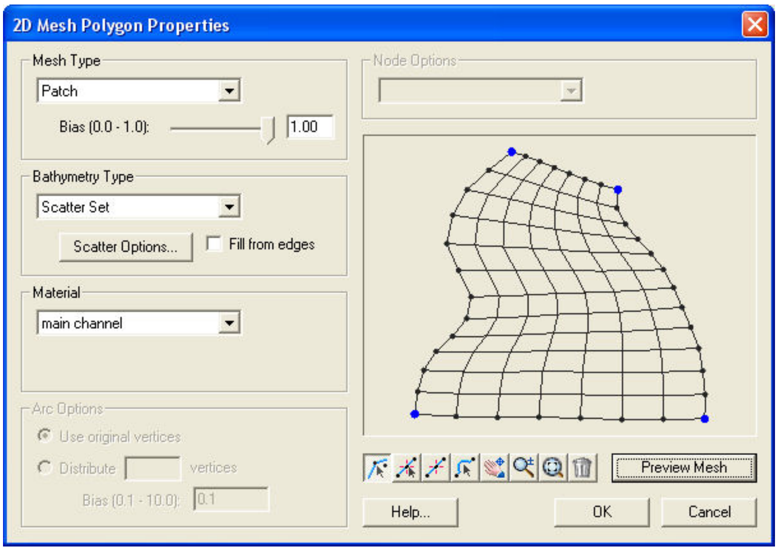
Related Topics

- Creating 2D Meshes
- Editing 2D Meshes
- 2D Mesh Generation
- Mesh Data Menu
- Mesh Module Display Options
- SMS Modules

3.8.a. Mesh Generation

Mesh Generation

At a glance



2D Mesh Polygon Properties Dialog

- Generating a quality finite element mesh is central to using many SMS models
- Conceptual models make generating meshes easier
- Polygons can use a variety of meshing options to generate triangular or quadrilateral elements
- Polygons can be assigned bathymetry and material information that will be transferred with the mesh
- Scalar paving density generates elements with sizes based upon a size dataset allowing for smooth transitions and a large range of element sizes and is particularly useful for coastal and wave models.
- Datasets for scalar paving density can be user defined or generated using the data calculator, the create datasets command, or LTEA (linear truncation error analysis) (ADCIRC)

2D Meshes can be created in the following different ways in SMS:

Using a Conceptual Model

This method converts a conceptual model to a mesh using the Map → 2D Mesh command. This is the preferred method for mesh generation in SMS.

The mesh generation capability is based on feature objects.

Mesh From Coverage

Each polygon in a meshing coverage (such as TABS, FESWMS, ADCIRC or CGWAVE) includes a mesh type attribute. This attribute defines how SMS should generate nodes inside the polygon and connect them into elements. Individual polygons in the coverage may each utilize their own meshing type. SMS supports the following principal mesh type attributes:

* Patch

* Paving

* Scalar Paving Density

Mesh Generation Toolbox

The mesh generation toolbox is the first pass at a new approach to further automating the mesh generation process. Currently the toolbox includes a single option to generate ADCIRC meshes with automatic density variation. This tool is based on Local Truncation Error Analysis.

Manually Creating a 2D Mesh

In order to create a 2D Mesh in SMS you must have a set of 2D Mesh nodes. Elements can be created by using one of the create mesh element tools and then selecting the mesh nodes to create elements. A 2D Mesh can also be created by triangulating the nodes. The triangulation algorithm assumes that each of the vertices being triangulated is unique in the xy plane, i.e. no two points have the same xy location. Duplicate points can be removed by selecting Select/Delete Duplicate Nodes command from the Node menu.

A 2D Mesh can be created manually from the following steps:

1. Select the Create Nodes tool from the Tool Palette.
2. Create the nodes by clicking inside the Graphics Window at the xy coordinates where you want the vertex located. (To change the node location see Editing Node Coordinates)
3. Select a create element tool from the Tool Palette **OR** Select the Elements | Triangulate command from the Mesh menu to form triangular elements using a Delaunay triangulation [1].

Creating a 2D Mesh from Existing Geometry

TINs, 2D grids, 2D scatter points, and 3D meshes (in GMS) can all be converted to a 2D Mesh. This is accomplished by using the following commands:

- TIN → 2D Mesh
- 2D Grid → 2D Mesh
- 2D Scatter Points → 2D Mesh Nodes
- 3D Mesh → 2D Mesh

After using the *Scatter Points → Mesh Nodes* command you must triangulate the nodes to create the 2D Mesh.

A finite element mesh is defined as a network of triangular and quadrilateral elements constructed from nodes. SMS includes advanced tools to create finite element meshes from underlying bathymetry, meshing parameters and mesh domain limits.

Digitized or survey points can be imported to provide the bathymetry. This type of data is generally not appropriate for use as mesh nodes due to random location and distribution. In this case the data should be converted to scatter points. If the bathymetric points are to be directly used as nodes, the triangulate command can generate elements from the points.

The Map Module provides tools for defining the study area boundaries and features from which a finite element mesh can be created. SMS then interpolates the bathymetry data onto the mesh. This process is also described in Lesson 2 of the tutorials. The Mesh Module provides various Tools for manually editing the finite element mesh.

Related Links

- Mesh Module
- Scatter Data Interpolation
- Adaptive Tesselation
- Advancing Front Triangulation
- Patches
- Refine Points

References

[1] http://en.wikipedia.org/wiki/Delaunay_triangulation

Refine Attributes Dialog

The Refine Attributes dialog is used to set the attributes for a refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each refine point include:

- Refine point (checked = on)
  - Element size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

Related Topics

- Feature Objects Menu
- Mesh Generation

Refine Point Dialog

The Refine Point dialog is used to set the attributes for a refine point represented by a feature point in a Cartesian Grid model coverage. Attributes that can be specified for each refine point include:

- Refine grid in I direction(checked = on)
  - Base cell size – Specify the cell I size in the vicinity of the refine point.
- Refine grid in J direction(checked = on)
  - Base cell size – Specify the cell J size in the vicinity of the refine point.

Only refine points located within a grid frame are used when the Map → 2D Grid command is executed. Refine points are not available for all models, since some Cartesian Grid models require uniform cell sizes.

Related Topics

- Feature Objects Menu

2D Mesh Options Dialog

The Map → 2D Mesh command is used to create a 2D mesh using the feature objects in a 2D Mesh Coverage. When the Map → 2D Mesh command is selected, the 2D mesh Options dialog opens. A meshing polygon must have been defined prior to issuing this command. The attributes of the meshing polygon(s) are used to generate the 2D mesh.

The 2D Mesh Options dialog is used to set options for the 2D mesh generation process. The options include:

- **Delete existing mesh** – If checked, the existing mesh will be deleted. If not checked, the new mesh will merge with the existing mesh
- **Merge triangles after meshing** – If checked, triangular elements created during the mesh generation process will be merged into quadrilateral elements where possible.
- **Copy coverage before meshing** – Create a copy of the coverage before the mesh generation algorithm redistributes vertices along the feature arcs defining meshing polygons. Feature arcs are only redistributed when using the Scalar Paving Density mesh generation method.
- **Use area coverage**

Related Topics

- Mesh Generation
- Map Feature Objects Menu

2D Mesh Polygon Properties

The 2D Mesh Polygon Properties dialog is used to set meshing options for the conceptual model. See Mesh Generation for a discussion of the conceptual modeling approach.

The following polygon attributes can be set:

- **Mesh Type** – Specify the mesh generation algorithm. The following options are available:
  - Patch
  - Paving
  - Scalar Paving Density.
- **Bathymetry Type** – Specify the bathymetry source for assignment to the mesh. The following options are available:
  - Constant – assignes a constant elevation value to all nodes in the mesh
  - Scatter Set – interpolates elevation values from the specified scatter set
  - Existing Mesh – interpolates elevation values from an existing 2D mesh
- **Material** – material type to assign to elements created within the polygon
- **Arc Options** – Used to modify the feature vertices along the arc. The arc must be selected in the polygon preview window. The following options are available:
  - Use Original Vertices
  - Distribute Vertices – Change the number of vertices along the arc. A bias can be specified rather than distributing the vertices with a constant spacing.
- **Node Options** – When the mesh type is set to Patch, the node options are used to define the 3 or 4 sides for the patch mesh generation. If a feature point is selected in the polygon preview window, the following options are available:
  - Split – The mesh generation algorithm will treat the two arcs meeting at the feature point as separate sides.
  - Merge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a single side.

- Degenerate Edge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a "degenerate edge." When using a degenerate edge, the Patching algorithm will require only 3 sides to be defined. This option is only valid for meshes which allow triangular elements. Only one degenerate edge can be specified per feature polygon.

Related Topics

- Feature Objects
- Map Module Menus

Advancing Front Triangulation

With advancing front triangulation the polygon is filled in layer by layer. In previous versions of SMS, this has been referred to as Paving. That term is associated with a specific algorithm, so the terminology is being changed here. The process includes offsetting the polygon boundary to the inside of the polygon (or outside of an island), performing intersections on this new offset layer and redistributing the vertices along the offset arc. The process is performed repeatedly until the area is filled with triangles.

Boundary Spaced Advancing Front

In the advancing front methodology utilized by SMS, the new layer position comes from the spacing on the vertices along the current polygon. The advanced front is created by forming equilateral triangles. The vertices on the new arc are redistributed based on the spatial interpolation of the original boundary spacings. This option requires no further input from the users.

Scalar Advancing Front

SMS supports the option to control the spacing between layers of elements using a Size Dataset. This is the scalar advancing front method and requires the user to select a spatial dataset that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources. See the tutorials on mesh generation for CGWAVE and ADCirc for more information.

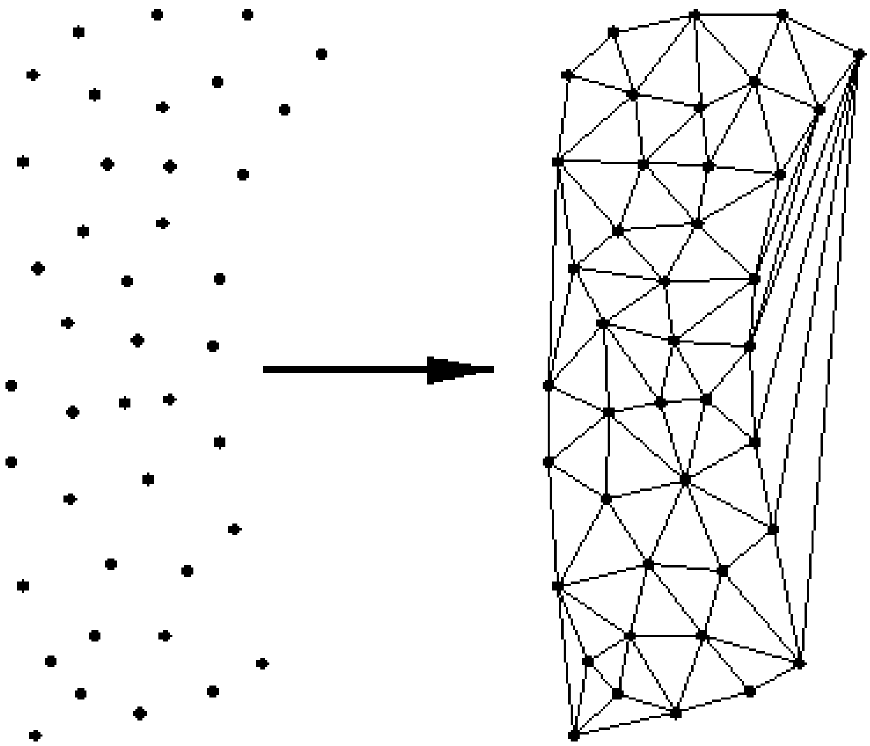
Related Topics

- Mesh Generation
- Adaptive Tesselation
- Patches

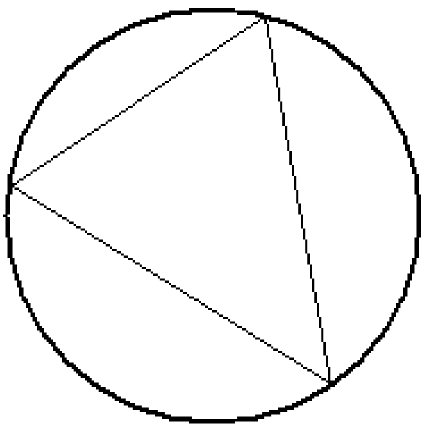
Mesh Node Triangulation

A simple means of creating a large number of elements is to triangulate a set of nodes into a network. This provides a surface that simulates the region being modeled, but normally does not result in quality elements.

This option is available through the Triangulate command from the Elements menu is executed. The selected nodes are connected with a series of triangles. If nodes are not selected, then all nodes will be triangulated. If linear elements exist, or a linear element creation tool has been selected, then this command creates linear triangles. Otherwise, quadratic triangles are created.



The triangulation algorithm ensures that the Delauney criterion is satisfied. The Delauney criterion is such that the circumcircle of a triangle does not enclose a node on any other element. The circumcircle of a triangle is the circle that passes through its vertices.



Optimize Triangulation

At times, the user will perform manual mesh editing using the Swap Edge tool. This makes the Delauney criterion no longer hold. Selected elements can be returned to the Delauney state by choosing the Optimize Triangulation command from the Elements menu.

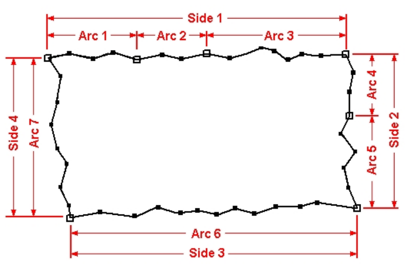
Related Topics

- Boundary Triangles
- 2D Mesh Elements Menu

Patch

The patch method fills polygons that can be defined as topologic rectangles. The method may combine multiple arcs to define a single side of the patch. A bidirectional coons patching method is used to interpolate from the boundary arcs to interior nodes. Typical applications of this method include river channels and regions aligned to channels. The patching method supports transitioning the number of elements across the channel or from one side to another. Transitions in both directions often result in poorer quality elements and should be avoided if possible.

The Rectangular Patch mesh generation method requires a polygon made of exactly four arcs, ***forming four sides***. However, very rarely do exactly four arcs make up a polygon. SMS provides a way to define a rectangular patch from a polygon that has more than four arcs by allowing multiple arcs to ***act as a single patch side***. An example of a rectangular patch made up of four sides is shown below. Note that Side 1 and Side 2 are both made from multiple arcs. Hollow squares represent the beginning and ending points of an arc (Feature Points). Filled squares represent intermediate points along the arc (Feature Vertices).



Definition of four sides for a rectangular patch

The basic process to define the meshing attributes for a polygon using the Patch method is to:

- Switch to the Map Module
- Select the menu Feature Objects | Build Polygons
- Switch to the Select Polygon tool
- Select the polygon you want to set meshing attributes for
- Select the menu Feature Objects | Attributes
- Set Mesh Type to Patch
- Set Bathymetry Type
- Set Material Type
- Set Node Options if the polygon consists of more than four arcs
  - Switch to the Select Feature Point tool in the dialog
  - Select the node you would like to "merge"
  - In the "Node Options" combo box, change the selection to "merge"

  - The two arcs meeting at the "merge" node will now be treated as a single arc for the mesh generation within the current polygon.
- Press "OK" to exit the dialog and save the polygon attributes

Related Topics

- Paving Meshing Algorithm
- Scalar Paving Density Meshing Algorithm
- Mesh Generation
- 2D Mesh Polygon Properties dialog

External Links

- Gonzales, Darren S. (2000). An Automatic Finite Element Mesh Generation Method: the Adaptive Rectangular Coons Patch. Thesis, Brigham Young University. TA 4.02 .G6476 2000

Patches

Patching is a mesh generation technique used to fill the interior of a polygon. A polygon is assigned to be a patch in the polygon attributes dialog and is filled with the Map → 2D Mesh command.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the polygon as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. Patches are applicable when the data points are gathered along parallel lines, such as cross sections in a river.

Rectangular Patches

The following are some hints when using rectangular patches:

The curvature of the patch can change somewhat, but it should not switch directions. If it does, then the patch should be split at the inflection point of the curve.

Although opposite sides in the rectangular patch are not required to have the same number of nodes, the best patches occur when this is close. In the example shown above, the two ends have the same number of nodes and the two sides only differ by three nodes.

Triangular Patches

All three sides of a triangular patch must have the same number of nodes.

Errors

When the patch is previewed in the polygon attributes dialog, the elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

If a polygon cannot be patched, a help string under the preview window in the polygon attributes dialog explains what needs to be changed.

Related Topics

- Mesh Generation
- Adaptive Tesselation
- Adaptive Front Triangulation

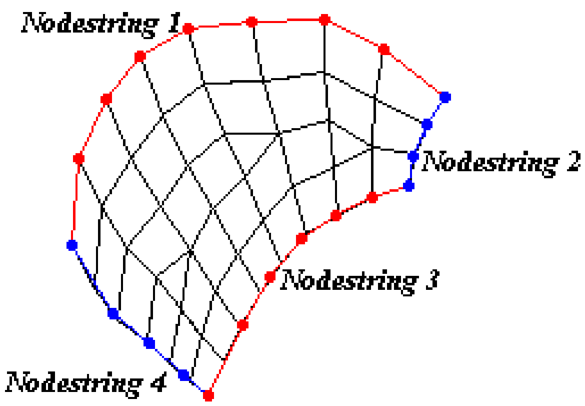
Paving

The paving method uses an advancing front technique to fill the polygon with elements. Based on the vertex distribution on the boundaries, equilateral triangles are created on the interior to define a smaller interior polygon. Overlapping regions are removed and the process is repeated until the region is filled. Interior nodal locations are relaxed to create better quality elements.

Related Topics

- Patch Meshing Algorithm
- Scalar Paving Density Meshing Algorithm
- Mesh Generation

Rectangular Patch



Elements can be made to fill a rectangular area by choosing the Rectangular Patch option from the Elements menu. To define a rectangular patch, four nodestrings must be selected. The nodestrings must connect at the ends.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the nodestrings as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. Patches are applicable when the data points are gathered along parallel lines, such as cross sections in a river. The following options are available for each edge of the rectangular patch:

- **Use original nodes** – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.
- **Distribute nodes** – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.
- **Bias** – This is used with the Distribute nodes option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

After the spacing on each side is defined, click the Preview button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the OK button to accept it. The patch can be canceled by clicking the Cancel button. Be careful to use the preview button because THERE IS NO UNDO FOR THIS OPERATION.

The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

The following are some hints when using rectangular patches:

- The curvature of the patch can change somewhat, but it should not switch directions. If it does, then the patch should be split at the inflection point of the curve.
- Although opposite sides in the rectangular patch are not required to have the same number of nodes, the best patches occur when this is close. In the example shown above, the two ends have the same number of nodes and the two sides only differ by three nodes.

Related Topics

- 2D Mesh Elements Menu

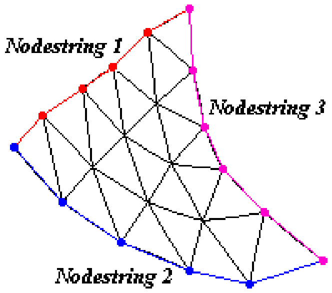
Scalar Paving Density

Scalar paving density utilizes the same approach as paving with the added component of a size dataset. A size dataset defines the desired spacing of nodes in a spatial fashion. A scattered dataset provides the geometric basis for the size dataset, and a dataset on the scatter set provides the values for the size dataset. SMS redistributes the vertices on the boundaries of the polygon to match the underlying size dataset.

Related Topics

- Patch Meshing Algorithm
- Paving Meshing Algorithm
- Mesh Generation

Triangular Patch



Elements can be made to fill a triangular area by choosing the Triangular Patch option from the Elements Menu. To define a triangular patch, three nodestrings must be selected. The nodestrings must connect at the ends.

The coordinates of the new nodes on the interior of the patch are computed by constructing a partial bicubic Coons patch using the nodestrings as patch edges. This ensures that interior nodes are smoothly interpolated from the nodes making up the perimeter of the patch. The following options are available for each edge of the triangular patch:

- Use original nodes – This option causes the original nodes from the nodestring to be used as corner nodes of elements along the boundary.
- Distribute nodes – This option distributes the specified number of nodes as corner nodes of elements along the boundary. If elements already exist on the boundary, then this option is unavailable.
- Bias – This is used with the Distribute nodes option. It causes the spacing of nodes along the nodestring to be weighted more to one of the corners.

All three sides of a triangular patch must have the same number of nodes. After the spacing on each side is defined, click the Preview button to see how the patch will look. If changes are desired, they can be made. When the patch looks good, click the OK button to accept it. The patch can be canceled by clicking the *CANCEL* button. Be careful to use the preview button because **THERE IS NO UNDO FOR THIS OPERATION**.

The elements in a new patch are checked to make sure they do not overlap each other. If any problems are detected, an error message is given and the patch is not created. Errors may occur especially when the region is highly irregular in shape. In such cases, the region can either be divided into smaller patches, or it can be filled using a different mesh generation technique.

Related Topics

- 2D Mesh Elements Menu

Adaptive Tesselation

Adaptive tessellation is a mesh generation technique used to fill the interior of a polygon. The method is based on overlaying a quad tree on the polygon, and recursively splitting the quads until the size approaches the desired spacing. SMS derives the desired spacing based on either the spacing of the original polygon, or based on a user specified spatially varying scalar dataset (for a scattered dataset). A polygon is assigned to be adaptive tessellation in the polygon attributes dialog and is filled with the Map to 2D Mesh command.

The adaptive tessellation technique is robust and relatively quick, however, it often results in discrete increments in resolution as the overlying quad tree grid transitions from one resolution level to another. For this reason, the Advancing Front Triangulation method is preferred.

Boundary Spaced Adaptive Tessellation

Adaptive tessellation uses the existing spacing on the polygons to determine the element sizes on the interior. Any interior arcs and refine points are forced into the new mesh. If the input polygon has varying node densities along its perimeter, SMS attempts to create a smooth element size transition between these areas of differing densities. By altering the size bias, the user can indicate whether SMS should favor the creation of large or small elements. Decreasing the bias will result in smaller elements; increasing the bias will result in larger elements. In either case, the elements in the interior of the mesh will honor the arc edges and the element sizes specified at nodes. The bias simply controls the element sizes in the transition region.

Scalar Adaptive Tessellation

SMS supports the option to control the local target size of elements using a Size Dataset. This is the scalar adaptive tessellation method and requires the user to select a spatial data set that is everywhere positive to define the local spacing of the desired mesh. This may come from a variety of sources. See the tutorials on mesh generation for CGWAVE and ADCirc for more information.

Related Topics

- Mesh Generation
- Advancing Front Triangulation
- Patches

3.8.b. Interface Components

Mesh Project Explorer Items

In the Project Explorer, the Mesh data folder houses all of the meshes that are contained in SMS. The Mesh data folder can hold as many meshes as the user desires, and can also generate sub folders. When a mesh is created in or read into SMS, a mesh item will be created under the Mesh Data folder. Meshes are considered 'active' when clicked on in the Project Explorer.

Mesh Module Right Click Menus

Right clicking on the Mesh Data folder in the *Project Explorer* will bring up the Display Options Dialog. Right clicking on a *Mesh Item* in the *Project Explorer* will bring up the following menu options:

- New Folder – Creates a new folder under the *Mesh* item
- Delete – Deletes mesh
- Convert – Converts mesh to Map or 2D Scatter.
- Projection – Allows the user to set the projection of the mesh.
- Reproject – Allows the user to reproject the projection of the mesh.
- Metadata – Allows the user to make annotations.
- Create Quality Mesh Scatter Set – This command creates a scatter set consisting of one vertex at the center of each element in the mesh. Six datasets are created for this scatter set including the six quality measures defined in the ARR quality plot. These datasets range from 0.0 to 1.0. The higher the value, the higher the quality of the element. (If SMS supported element centered datasets, these quantities could be displayed directly on the mesh.) The creation of this scatter set gives the user a spatial feeling for quality of the mesh.
- Zoom to Mesh – Zooms to where the mesh is located within the *Graphic Window*.

3.8.b.1. Mesh Module Display Options

Mesh Module Display Options

The properties of the mesh data SMS displays on the screen can be controlled through the Display Options dialog. The entities associated with the mesh module with display options are shown below. Some of these entities also have an associated Options button. For these entities, additional display options are available. The available mesh display options include the following:

- **Nodes –** A circle is filled around each node. The user can specify the radius and color of these circles. The Options button is used to set the display of nodal boundary condition data. The dialog that opens when this button is clicked depends on the current numerical model.
- **Nodal BC –** Some models support nodal boundary conditions. For those that do, each type of boundary condition can be displayed by highlighting the node with a symbol associated with that boundary condition. The Options button next to the Nodal BC entry of the display options allows the user to select/modify the symbols associated with each boundary condition.
- **Elements –** Element edges.
- **Functional Surface**
- **Contours –** The mesh contours are drawn for the active scalar data set. Use the contours tab to change contour options.
- **Vectors –** The mesh vectors are drawn for the active vector data set. Use the vectors tab to change vector options.
- **Nodestrings –** The color in which a nodestring is drawn depends upon its type. Unassigned nodestrings are drawn in the color/thickness/style shown at the left of the toggle box. For the display of boundary condition nodestrings, click the Options button. The dialog that opens when this button is clicked depends on the current numerical model.
- **Mesh Quality –** The mesh quality shows potential problems with the finite element mesh layout. An element is highlighted in a color corresponding to the criterion which it violates. The Options button opens the Mesh Quality Options dialog to specify display options for the mesh quality criteria.
- **Mesh Boundary –** A line is drawn around the perimeter of the mesh.
- **Materials –** Elements can be filled with the color and pattern which define their materials. Materials and their display properties can be edited by choosing Edit|Materials from the menu.
- **Material Boundary –** The boundary between zones of elements with a common material type is drawn using specified line attributes.
- **Material Numbers –** The material id number can be displayed in the center of each element. User selects font and color.
- **Node Numbers –** The node id number can be displayed next to each node. User selects font and color.
- **Element Numbers –** The id of each element.
- **Nodal Elevations –** Displays the z elevation at each node.
- **Wet/Dry Boundary –** After a simulation has been opened, the interface between wet and dry nodes can be displayed.

Model specific options

Each model may include model specific display options. These appear at the bottom of the display options dialog and include such things as 1D contour options for RMA2 and tidal ellipses for ADCIRC.

Related Topics

- Mesh Module
- Display Options

Mesh Quality

Several rules of mesh element construction, if adhered to, will help in creation of a well-behaved finite element network. Violations of the following mesh quality checks should be avoided. Violations of these mesh quality checks can be displayed in SMS (see Mesh Display Options):

- **Minimum / Maximum interior angle –** For triangular elements, if the angle is between 10 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- **Concave quadrilaterals –** For quadrilateral elements, if the angle is between 30 and 150 degrees, computation problems will usually be avoided. Care must also be taken when curved edges are defined (non-linear midside nodes) to prevent overlap of element sides.
- **Maximum slope –** Rapid changes in slope can cause computational instabilities.
- **Element area change –** Nodes need to be more plentiful and elements smaller in areas where the solution variables (u,v, and h) change rapidly. Such areas may be located near channel or floodplain constrictions, in channel bends, or at sudden changes in bed slope. The network should be dense in the critical areas of interest. The density of a network can vary through the solution domain. Areas that are of little interest and have stable flow characteristics should not be as dense as critical areas. The size of elements needs to change gradually when moving from an area described by small elements to an area modeled with large elements, or vice versa. A rule of thumb is to keep the areas of neighboring elements within a factor of two, meaning an element is twice as big or half as big as its adjacent elements.
- **Connecting elements –** Avoid creating "pinwheels" by limiting the number of elements connecting at a node to fewer than eight.
- **Ambiguous gradient –** All triangular elements are planar by their definition. However, quadrilateral elements may vary significantly from a plane. It is a good idea to construct elements as close to a plane as possible. This precludes the existence of elements whose slope, or direction of drainage is ambiguous.

Related Topics

- Mesh Display Options
- Mesh Module
- Display Options

3.8.b.2. 2D Mesh Module Tools

2D Mesh Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. Tools specific to a model interface are described with the corresponding model. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the 2D Grid tool palette.

Tool

Tool Name Create Mesh Nodes Select Mesh Nodes Create Nodestrings Select Nodestrings Create Elements Select Elements Swap Edges Merge/Split Elements Label Contours Description The Create Mesh Nodes tool is used to manually create a node using the mouse. A node will be created at the location where the mouse button is clicked inside the Graphics Window. If the node is created inside the triangulated area of a current mesh, and the Insert nodes into triangulated mesh option is turned on, then the new node will be added as part of the mesh. If the new node is not added as part of the current mesh, then the z-value assigned depends on the Nodal z-value option.

The Select Mesh Nodes tool is used to select nodes. A single node is selected by clicking on it. A second node can be added to the selection list by holding the SHIFT key while selecting it. Multiple nodes can be selected at once by dragging a box around them. A selected node can be deselected by holding the SHIFT key as it is clicked.

If the nodes are not locked (see the menu command Nodes | Locked), then a single node can be clicked and dragged to a new location. As the node is being dragged, its new location is shown in the Edit Window. If a single node selected, the X, Y, and Z Coordinate fields in the Edit Window become available to set the node location exactly. If multiple nodes are selected, the Z Coordinate field in the Edit Window becomes available. The value shown is the average elevation value of all selected nodes. If this value is changed, the new value will be assigned to all selected nodes.

With one node selected, the Edit Window shows the node id number and the number of elements to which it is attached. With two nodes selected, the Edit Window shows both node id numbers and the distance between the nodes. With multiple nodes selected, the Edit Window shows the number of selected nodes.

The Create Nodestrings tool is used to create node string. Nodestrings are used for operations such as assigning boundary conditions, forcing breaklines into the mesh, and renumbering the mesh. To create a nodestring:

1. Click on a node. The node will be highlighted in red and a prompt will be shown in the Help Window.
2. Click on any node to add it to the nodestring. The selected node is also highlighted in red and a solid red line is drawn between the two nodes. Continue adding nodes to the nodestring in this manner.

    1. Note: For most operations, nodes in the nodestring should be adjacent, but this is not required. A breakline, for example, will usually be made of nodes which are not adjacent.
    2. Press the BACKSPACE key to backup one node. Press the ESC key to abort the nodestring creation.
    3. Double-click a node or press the ENTER key to end the nodestring creation.
3. The SHIFT and CTRL keys assist in creating large nodestrings which are made up of adjacent nodes. These can be used after at least one node has been selected and function as follows:
    1. SHIFT. Holding down the SHIFT key and selecting another node will add to the nodestring all nodes between the two. The path chosen is the shortest distance between the two nodes. This is useful for creating continuity strings which run along a cross section of the mesh.
    2. CTRL. Holding down the CTRL key and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going counter clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.
    3. CTRL + SHIFT. Holding down both the CTRL and SHIFT keys and selecting another node will add to the nodestring all nodes on the mesh boundary between the two, going clockwise from the first node to the second node. Both nodes must be on the boundary of the mesh or SMS will beep.

The Select Nodestrings tool is used to select nodestrings. When this tool is chosen, a small icon appears near the center of each nodestring. A nodestring is selected by clicking inside this icon. A second nodestring can be added to the selection list by holding the SHIFT key while selecting it. Multiple nodestrings can be selected by dragging a box around their icons. A selected nodestring can be deselected by holding the SHIFT key as its icon is clicked.

When nodestrings are selected, the Z Coordinate field in the Edit Window becomes available. The value shown is the average elevation value of all nodes in the selected nodestrings. If this value is changed, the new value will be assigned to all nodes in the selected nodestrings.

With one nodestring selected, the Edit Window shows the number of nodes in the nodestring, its type, and its length. With multiple nodestrings selected, the Edit Window shows the number of selected nodestrings and their total length.

Most elements in SMS will be created using automatic mesh generation techniques. At times, however, it is necessary to manually create a single element or a small group of elements.

Although SMS supports various types of elements, only those element types supported by the current numerical model will be available in the tool palette. Some of these element types are linear while others are quadratic. A linear element has only corner nodes, while a quadratic element has midside nodes between the corner nodes.

The following linear elements are supported:

2-node lines 3-node triangles 4-node quadrilaterals.

The following quadratic elements are supported:

3-node lines 6-node triangles 8-node quadrilaterals 9-node quadrilaterals.

Linear and quadratic elements cannot coexist in a single mesh. If linear elements exist in a mesh, then the quadratic element creation tools are dimmed out. Similarly, if quadratic elements exist in a mesh, then the linear element creation tools are dimmed out. To create a single linear or quadratic element:

1. Select the tool which corresponds with the type of element to be created.
2. Click on the corner nodes which will make the element, one-by-one. Do not click midside nodes. As each node is clicked, it becomes highlighted in red.
3. Alternatively, a box can be dragged around the corner nodes which will make the element. A beep will sound if the box does not surround the exact number of corner nodes required by the selected tool.

The midside nodes in quadratic elements are created automatically, as is the center node of a nine-node quadrilateral. Before a new element is created, SMS performs the following quality checks:

- The new element cannot overlap other elements.
- A quadrilateral element cannot twist or overlap itself.
- A quadrilateral element cannot be concave.

If any of the these fails, the new element is not created.

The Select Elements tool is used to select elements. A single element is selected by clicking inside it. A second element can be added to the selection list by holding the SHIFT key while selecting it. Multiple elements can be selected at once by dragging a box around them. Holding the CTRL key and dragging the mouse selects any elements through which the line is drawn. A selected element can be deselected by holding the SHIFT key as it is clicked.

When elements are selected, the Z Coordinate field in the Edit Window becomes available. The value shown is the average elevation value of all nodes in the selected elements. If this value is changed, the new value will be assigned to all nodes attached to the selected elements. Caution must be used when changing node elevations in this manner. Do not create large flat areas where surrounding elements may become dry because this can cause ponds to form when the finite element analysis is performed.

With one element selected, the Edit Window shows the element id number, its type, and its area. With multiple elements selected, the Edit Window shows the number of selected elements and their combined area.

The Swap Edges tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as preserving a geometrical feature in the mesh or avoiding a artificial dam in a channel.

Two adjacent triangles form a quadrilateral with an element edge down one diagonal. When the diagonal is clicked, it gets swapped to the other diagonal, as long as the quadrilateral is not concave.

The Merge/Split Elements tool is used to either merge two triangles into a quadrilateral or split a quadrilateral into two triangles. This is a useful tool to use when trying to avoid certain mesh drying problems.

To split a quadrilateral element, click inside it. An element edge appears on the diagonal which will make the triangles uphold the Delaney criteria. To merge two adjacent triangles into a quadrilateral, click the common edge. A quadrilateral will form as long as it is not concave.

The Label Contours tool is used to manually create a contour label using the mouse. To add a label, click on the point where the label should be created. The label will remain on the screen until either it is manually removed or the automatic contour label options are changed. To manually remove a contour label, hold the SHIFT key and clicking on it. There are also available automatic contour label options. Right Click Menu N/A See the 2D Mesh Nodes Menu and 2D Mesh Elements Menu articles. N/A See the 2D Mesh Nodestrings Menu article. N/A See the 2D Mesh Elements Menu article. N/A N/A N/A

Related Topics

- Editing 2D Meshes
- Mesh Module

Editing 2D Meshes

2D Mesh nodes can be inserted, deleted, or moved.

2D Mesh elements can be edited in the following ways:

- Elements can be converted between linear and quadratic.
- The type of element can be changed from a 3 node element to a 4 node element by merging triangles.
- A 4 node element can be converted to a 3 node element by splitting the 4 node element.
- Elements can be refined automatically.
- The material assigned to an element can be changed.
- Poorly shaped boundary triangles can be automatically selected for deletion.
- Breaklines can be inserted into the mesh

Deleting Nodes

A set of selected nodes can be deleted by hitting the Delete key or selecting the Delete command from the Edit menu. Elements attached to the nodes are also deleted.

If the Retriangulate voids when deleting option is turned on, the void created when a node and the elements surrounding the node are deleted is re-triangulated or filled in with triangles. This feature makes it possible to selectively "unrefine" a region of the mesh or reduce the density of the nodes in a region of the mesh without having to completely recreate all of the elements in the region.

If the Retriangulate voids when deleting item in the Node Options dialog is not set, the selected node and the elements surrounding the node are simply deleted and the resulting void is not filled in with triangles.

If the Confirm Deletions option in the Edit menu is active, GMS will prompt the user to confirm each deletion. This feature is helpful in preventing accidental deletions. The Confirm Deletions item is toggled by selecting it from the menu.

Editing Node Coordinates

The coordinates of a 2D Mesh node can be edited by selecting the mesh node and entering the new coordinates in the edit boxes in the Edit Window. It is also possible to drag an existing node to a new location by clicking on the node and moving the mouse with the button held down until the node is in the desired position.

If the snap to grid option in the Drawing Grid Options dialog is set, the node will move in increments corresponding to the drawing grid. If the node being dragged is connected to one or more elements, SMS will not allow the node to be dragged to a position where one of the surrounding elements would become ill-formed.

Since it is possible to accidentally drag points, nodes can be "locked" to prevent them from being dragged by selecting the Lock All Nodes item from the Nodes menu. The nodes can be unlocked by unselecting Lock All Nodes from the Nodes menu.

Merging Triangles

The triangulate operation creates a mesh composed entirely of triangles. In some cases it is desirable to have the mesh composed primarily of quadrilateral elements. Quadrilateral elements result in less elements which leads to faster solutions, and quadrilateral elements are often more stable numerically. To address this need, two options are provided for converting triangular elements to quadrilateral elements:

The Merge Triangles Command

The Merge Triangles command in the Elements menu can be used to automatically merge pairs of adjacent triangular elements into quadrilateral elements. Upon selecting the Merge Triangles command, the user is prompted to input a minimum interior angle. This angle should be between 0 and 90 degrees. If no elements are selected, all of the triangular elements in the mesh are then processed. If some elements have been selected, only the selected elements are processed.

The conversion process works as follows:

1. The set of elements to be processed is traversed one element at a time. Each triangular element that is found is compared with each of its three adjacent elements. If the adjacent element is a triangle, the trapezoid formed by the triangle and the adjacent triangle is checked.
2. Each of the four interior angles of the trapezoid is computed and compared to a minimum interior angle. If all of the angles are greater than the user-specified minimum interior angle, then the two triangles are merged into a single quadrilateral element.

This process is repeated for all of the elements. The merging scheme will not always result in a mesh composed entirely of quadrilateral elements. Some triangular elements are often necessary in highly irregular meshes to provide transitions from one region to the next.

The Merge/Split Tool

The other option for merging triangles involves the use of the Merge/Split tool in the Tool Palette. This tool can be used to manually merge triangles one pair at a time rather than using the automatic scheme described above. The manual method is also useful to edit or override the results of the automatic merging scheme in selected areas. The Merge/Split tool can also be used to undo a merge. A quadrilateral element can be split into two triangles by clicking anywhere in the interior of the element. This tool is useful if a pair of triangles is inadvertently merged.

Splitting Quadrilaterals

Occasionally it is necessary to split quadrilateral elements into triangular elements. For example, in order for new nodes to be automatically inserted into a mesh, the elements in the region where the node is inserted must be triangular. Also, in order to process a breakline, the elements in the region of the breakline must be triangular. In such situations, it may be necessary to split a group of quadrilateral elements into triangular elements. Two options are provided for splitting quadrilateral elements:

The Split Quads Command

The Split Quads command in the Mesh menu can be used to split a group of quadrilateral elements into triangular elements. If no elements are selected, all of the quadrilateral elements in the mesh are split. If some elements have been selected, only the selected quadrilateral elements are split.

The Merge/Split Tool

The other option for splitting quadrilateral elements involves the use of the Merge/Split tool in the Tool Palette. If the Merge/Split tool is selected, clicking anywhere in the interior of a quadrilateral element with the mouse cursor will cause the element to be split into two triangles. The shortest diagonal through the quadrilateral is chosen as the common edge of the two new triangular elements.

Mesh Subset Edit Mode

When working with large meshes, it can be useful to work on only a small portion of the mesh. The user can select elements on which to work on and enter subset edit mode either through the select element tool's right click menu or the right click menu on the mesh tree item. The geometry of the mesh inside the selected area can then be changed. Datasets other than the depth/elevation dataset are deleted upon entering mesh subset edit.

Prohibited Actions While in Subset Edit Mode

- Making changes on the border of the mesh subset
- Making changes outside of the mesh subset
- Reading in meshes from file
- Creating meshes from feature map or scatter data
- Running models
- Changing attributes such as boundary conditions on nodes, nodestrings, and elements
- Renumbering node/nodestring/element ids

Commit Mesh

The user can merge the changes made to the subset of the mesh with the rest of the mesh by selecting Commit mesh from the right click menu of either the mesh tree item or the select element tool. Commit mesh will exit subset edit mode.

Revert Mesh

The user can revert to how the mesh was upon entering mesh subset edit mode by selecting Revert mesh from the right click menu of either the mesh tree item or the select element tool. Revert mesh will exit subset edit mode.

Related Topics

- 2D Mesh Module Tools

Mesh Nodestring Boundary Conditions Dialog

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary. The options available will be based on the current generic model in use and are defined in the Define Model Dialog.

Related Topics

- Generic 2D Mesh Arc Attributes Dialog

2D Mesh Module Tools Right Click Menus

The following tools are contained in the Dynamic Tools portion of the tool palette when the Mesh Module is active. These are tools with corresponding right-click menus.

Tool	Tool Name	Right-click Menu
	Select Mesh Nodes	• Delete – Delete selected mesh Nodes. • Assign BC – Assign a boundary condition to selected mesh nodes. • Transform – Transform the selected nodes either by scaling, translation, or rotation. • Triangulate – Triangulate the selected nodes to form triangle elements.
	Select Nodestrings	• Delete Selected – Delete selected nodestrings. • Assign BC – Assign a boundary condition to selected nodestrings. • Reverse Direction – Reverse direction of selected nodestrings. • Force Breaklines – Force element edges to follow the selected nodestrings. • Renumber Nodes – Renumber the nodes starting with an ID of 1. • Renumber Nodestrings – renumber the nodestrings starting with an ID of 1. • Smooth – Smooth the mesh boundary along the path of a selected nodestring by moving midside nodes. Only used for quadratic elements.
	Select Elements	• Delete – Delete selected element. • Refine – Split all selected elements into four elements. • Relax – Moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command. • Assign Material Type – Assign material type to selected elements.

Related Topics

- Editing 2D Meshes
- Mesh Module

3.8.b.3. 2D Mesh Module Menus

2D Mesh Module Menus

The following menus are available in the 2D Mesh Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data
- Nodes
- Nodestrings
- Elements

Right Click Menu

In addition to the standard right click menu items (Project Explorer) the following menu item appears:

- **Create Mesh Quality Scatter Set –** This command creates a scatter set consisting of one vertex at the center of each element in the mesh. Sixdata sets are created for this scatter set including the six quality measures defined in the ARR quality plot. These datasets range from 0.0 to 1.0. The higher the value, the higher the quality of the element. (If SMS supported element centered datasets, these quantities could be displayed directly on the mesh.) The creation of this scatter set gives the user a spatial feeling for quality of the mesh.

Model Specific Menus

- ADCIRC
- ADH
- CGWAVE
- FESWMS
- Generic Model
- TABS
  - RMA2
  - RMA4

2D Mesh Nodestrings Menu

The Nodestrings Menu includes the following commands:

General Commands

Command	Description
Options	breakline options
Force Breaklines	Force element edges to follow the selected nodestring
Smooth	Smooth the mesh boundary along the path of a selected nodestring by moving midside nodes. Only used for quadratic elements.
Renumber Nodestrings	Renumbers the nodestrings starting with an id of 1.
Merge	Merge selected nodestrings to form a single nodestring. Available if more than one nodestring is selected.
Split	Split a single nodestring into multiple nodestrings at the selected mesh node. Available if a mesh node is selected.
Reverse Direction	Reverse the direction of selected nodestrings. Selecting a nodestring causes the direction arrows to be displayed and can be used to verify the nodestring direction.

Nodestring Options

Breakline Options

Controls how breaklines are processed.

- **Insert new nodes –** Triangles intersected by the breakline are modified by adding new nodes at necessary locations to ensure that the edges of the triangles will conform to the breakline. The elevations of the new nodes are based on a linear interpolation of the breakline segments. The locations of the new nodes are determined in such a way that the Delauney criterion is satisfied.
- **Swap element edges –** Triangles intersected by the breakline are modified by swapping element edges to ensure that the edges of the triangles will conform to the breakline.

Related Topics

- Mesh Module

Mesh Data Menu

Most of the SMS modules have a Data menu, but the items in this menu are different for each module. The Mesh Module commands include:

Command	Description
Steering Module	Model command – launches the steering dialog to connect multiple model runs
Switch Current Model	Model command – allows the user to change the current numerical model associated with the mesh
Data Calculator	Dataset command – invokes the data calculator for the mesh
Create Datasets	Dataset command – creates specific datasets for the mesh based on user specified options
Map Elevation	Dataset command – assigns a new "elevation" or "depth" dataset to the current mesh. See below.
Zonal Classification	Dataset command – creates polygons matching user specified criterion from the current mesh and its datasets
Contour Options	Visualization command – invoke the contour options dialog (display options)
Vector Options	Visualization command – invoke the vector options dialog (display options)
Film loop	Visualization command – launches the film loop generation wizard
Mesh → Scatterpoint	Data conversion command – converts the current mesh to a scatter point set. Also available as a right click command. See Below.
Mesh → Map	Data conversion command – converts the current mesh to a map module coverage. Also available as a right click command. See Below.

Map Elevation

SMS requires that a mesh or grid have bathymetry, or bottom elevation data, associated with the nodes or cells. By default, SMS creates a dataset named "elevation" to store the elevation values. The dataset being used to store elevations is referred to as the mapped dataset.

The Data menus in several modules include a command to use another functional dataset as this mapped elevation function. When this command is performed the Select Dataset dialog opens to allow any existing scalar dataset to be chosen. Any time step of any scalar dataset can be used as the mapped dataset and override the previous nodal elevation values. This is used mainly when interpolating new elevation data from scatter points.

Mesh to Map

The Mesh → Map command in the data menu (mesh module) is used to convert mesh data into feature data (map module). This can be useful for creating a conceptual model from an existing numeric model. The Mesh → Map command converts the mesh data and places it in the active coverage. If you want the new data in a new coverage, click on the *Create New Coverage* button to create a new coverage and make it active.

Material Regions → Polygons

This option converts the materials in the current mesh into polygons in the map module. If the coverage supports materials (area property or most mesh model coverages support this), the polygons will have attributes reflecting the correct material values.

Mesh Boundaries → Polygons

This option converts the current mesh boundaries into polygons in the map module.

Mesh Contours → Arcs

This option creates an arc at a specific contour value (based upon the active dataset). It is not necessary for the value to be associated with a displayed contour line as SMS will determine where the line would be if it existed.

Mesh Nodestrings → Arcs

This option creates an arc for each nodestring in the mesh.

Mesh to Scatterpoint

The Mesh → Scatterpoint command in the Mesh Data menu is used to create scatter point data from existing mesh nodes. When this command is performed, a prompt appears to request a name for the new scatter point set.

When the scatter point set is created, it contains one scatter point for each mesh node, including midside nodes and center nodes. Any mesh datasets that have been read into SMS are copied into the new scatter point set. The scatter point data can then be used for interpolation.

Related Topics

- Mesh Module

3.8.b.3.1. 2D Mesh Elements Menu

2D Mesh Elements Menu

The Elements Menu includes the following commands:

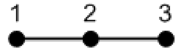
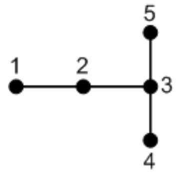
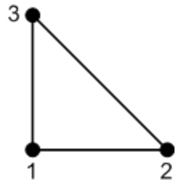
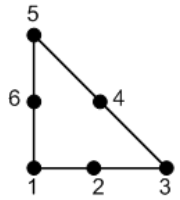
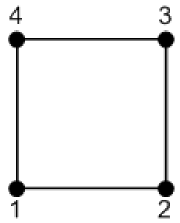
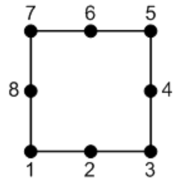
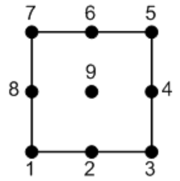
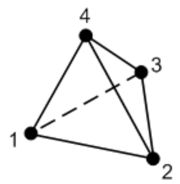
Command	Description
Options	General command – set up the default element options.
Select Thin Triangles	General command – selects all thin triangular element that meet the definition specified in the options command.
Find	General command – finds the element with a specified ID or location.
Assign Material Type	General command – requires a selected element. Sets the material type of the selected elements based on the option defined in the options command.
Merge Triangles	Conversion command – merges triangle pairs that meet the standard for rectangles defined in the options command. Can operate on selected elements.
Split Quadrilaterals	Conversion command – splits quadrilateral elements into two triangular elements. Can operate on selected elements.
QUAD8 ↔ QUAD9	Conversion command – converts all QUAD8 to QUAD9 elements and vice versa. Only applies to the FESWMS model. Can operate on selected elements.
Linear ↔ Quadratic	Conversion command – converts all elements from linear to quad or vice versa. Only applies to TABS and FESWMS models.
Triangulate	Generation command – triangulates the selected nodes to form triangle elements.
Optimize Triangulation	Generation command – swaps edges of triangular elements to meet the Dulanay criterion.
Refine	Generation command – splits all selected elements into four elements.
Relax	Generation command – moves all the nodes on the interior of the selected elements to the centroid of their contributing area. Iterates based on option set in the options command.
Fix Bad Area Transitions	Generation command – removes nodes that cause bad area transitions as defined in the element quality control.
Rectangular Patch	Patch command – creates elements from four selected node strings.
Triangular Patch	Patch command – creates elements from three selected node strings.

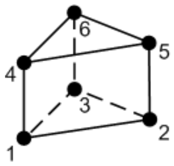
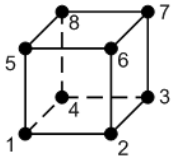
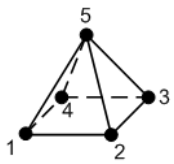
Related Topics

- Mesh Module

Element types

Element types used in XMS software. See also XMDF elements [1].

<table>
<tr><th>Element Type</th><th>Image</th><th>Faces</th></tr>
<tr><td>1D linear element with 2 nodes</td><td>
</td><td></td></tr>
<tr><td>1D linear element with 3 nodes</td><td>
</td><td></td></tr>
<tr><td>transition element</td><td>
</td><td></td></tr>
<tr><td>2D linear triangle</td><td>
</td><td></td></tr>
<tr><td>2D quadratic triangle</td><td>
</td><td></td></tr>
<tr><td>2D linear quadrilateral</td><td>
</td><td></td></tr>
<tr><td>2D quadratic quadrilateral</td><td>
</td><td></td></tr>
<tr><td>2D quadratic quadrilateral with center node</td><td>
</td><td></td></tr>
<tr><td>3D linear tetrahedron</td><td>
</td><td><table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>2,3,4</td></tr>
<tr><td>2</td><td>1,4,3</td></tr>
<tr><td>3</td><td>1,2,4</td></tr>
<tr><td>4</td><td>1,3,2</td></tr>
</table></td></tr>
</table>

<table>
<tr>
<td>3D linear prism</td>
<td>
</td>
<td>
<table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,3,2</td></tr>
<tr><td>2</td><td>4,5,6</td></tr>
<tr><td>3</td><td>1,2,5,4</td></tr>
<tr><td>4</td><td>2,3,6,5</td></tr>
<tr><td>5</td><td>3,1,4,6</td></tr>
</table>
</td>
</tr>
<tr>
<td>3D linear hexahedron</td>
<td>
</td>
<td>
<table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,4,3,1</td></tr>
<tr><td>2</td><td>5,6,7,8</td></tr>
<tr><td>3</td><td>1,2,6,5</td></tr>
<tr><td>4</td><td>2,3,7,6</td></tr>
<tr><td>5</td><td>3,4,8,7</td></tr>
<tr><td>6</td><td>4,1,5,8</td></tr>
</table>
</td>
</tr>
<tr>
<td>3D linear pyramid</td>
<td>
</td>
<td>
<table>
<tr><th>FaceID</th><th>Node Indices</th></tr>
<tr><td>1</td><td>1,4,3,2</td></tr>
<tr><td>2</td><td>1,2,5</td></tr>
<tr><td>3</td><td>2,3,5</td></tr>
<tr><td>4</td><td>3,4,5</td></tr>
<tr><td>5</td><td>4,1,5</td></tr>
</table>
</td>
</tr>
</table>

Back to XMS

References

[1] http://www.aquaveo.com/xmdf/doc1.6/group__d4d8d2d.html

Assign Material Type

Each element in the mesh is assigned a material type. The default material ID can be set in the *Element Options* dialog. A selected element is assigned a new material type by choosing the *Assign Material Type* command from the Elements Menu. If the Assign default material option is selected in the *Element Options* dialog, then the default material is automatically assigned to the selected element. If the Prompt for material when assigning option is selected in the *Element Options* dialog, then the *Materials Data* dialog opens from which a material type can be chosen.

Related Topics

- 2D Mesh Elements Menu

Boundary Triangles

The perimeter of the mesh resulting from the triangulation process corresponds to the convex hull of the data points. This may result in some long thin triangles or "slivers" on the perimeter of the triangulated region.

Select Thin Triangles

There are several ways to select and delete long thin triangles.

Long thin triangles on the perimeter of the mesh can be automatically selected using the Select Thin Triangles item from the Elements menu. The triangles on the outer boundary are checked first and if the aspect ratio of a triangle is less than a critical value, the triangle is selected and the triangles adjacent to the triangle are then checked. The process continues inward until none of the adjacent triangles violate the minimum aspect ratio.

The "drag line" method for selecting elements was designed specifically for this purpose. Elements can be selected with a line by selecting the Select Elements tool, holding down the Control key, and dragging a line through all of the elements to be selected. The selected elements can then be deleted.

Related Links

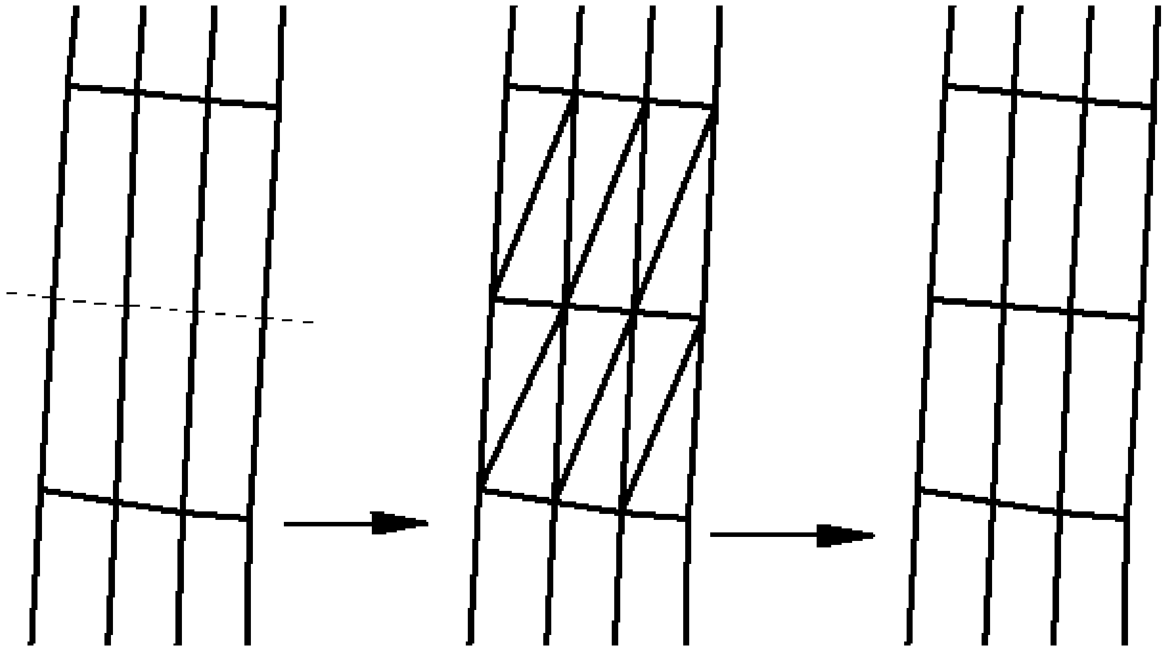
- Editing 2D Meshes
- Mesh Node Triangulation

Convert Elements

Several commands are available to modify the current elements these include:

Add Breakline

The Add Breaklines command from the Elements menu can be executed when at least one nodestring has been selected. This forces element edges along the nodestring line. When this command is performed, elements are sliced along the nodestring to ensure that the edges will conform to the breakline. The elevations of any new nodes are interpolated from the original mesh. All new triangles satisfy the Delauney criterion.



A breakline example is shown. This example has some long, skinny quadrilaterals which will be split across the width. The dotted line in the left part of the figure represents the location of the breakline. When the elements are split, triangles are formed. These can be merged together using the Split/Merge tool , as shown in the right part of the figure.

Merge Triangles

The Split/Merge tool can be used to merge individual pairs of triangles. Doing this manually for large numbers of elements takes a lot of time. The Merge Triangles command from the Elements menu automatically merges a selected set of triangles simultaneously. If no elements are selected when this command is executed, all triangles in the finite element mesh will be processed.

This command uses the Merge triangles feature angle specified in Element Option dialog. In order to form quadrilateral elements with the best aspect ratios, SMS starts with a feature angle of ninety degrees and checks for any elements that can be merged. Then, a series of steps are performed, each time lowering the feature angle and checking for elements that can be merged. This ensures that the quadrilaterals which are formed are as close to rectangular as possible. In general, after the automatic merging process is complete, a limited number of triangles will still exist.

Split Quadrilaterals

The Split Quadrilaterals command in the Elements menu is used to split a set of quadrilaterals into triangles. If no elements are selected, all quadrilateral elements in the mesh will be split. The quadrilaterals are split along the shortest diagonal.

Quad8 ↔ Quad9

The Quad8↔Quad9 item from the Elements menu is used to convert between eight- and nine- noded quadrilaterals. FESWMS supports nine-noded quadrilaterals. Both FESWMS and TABS support eight-noded quadrilaterals. If no elements are selected when this command is performed, all elements are converted.

Linear ↔ Quadratic

Linear elements (three node triangles and four node quadrilaterals) can be converted to quadratic elements (six node triangles and eight node quadrilaterals) and vice versa by selecting the Linear ↔ Quadratic item from the Elements menu. A finite element mesh must be made of either all linear elements or all quadratic elements. Linear elements do not have midside nodes while quadratic elements do.

Refine

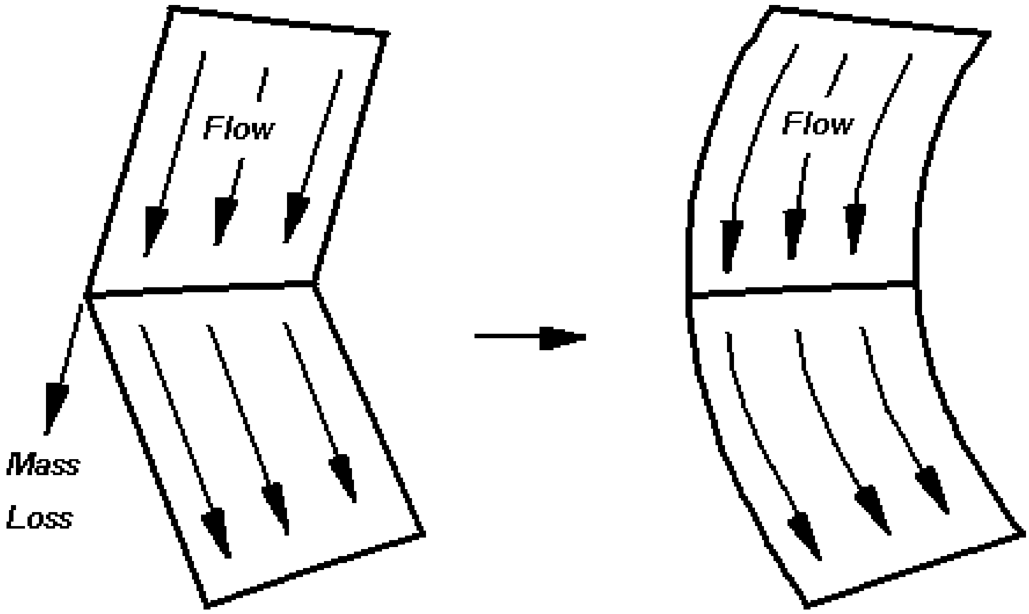
At times, there is not enough definition in a finite element mesh. The Refine command from the Elements menu splits each of the selected elements into smaller elements. After the selected elements have been refined, SMS automatically creates transitions, from the refined area of higher density to the unrefined area of lower density, using triangular elements. Refine options are set in the Element Options Dialog.

Relax

See the article on mesh relaxation.

Smooth Nodestring

Quadratic elements have a node located at the midpoint of each edge. These nodes are generally referred to as midside nodes. The angular corners resulting from such elements are discontinuous. Such a discontinuity may result in inaccuracy in the numerical model sometimes referred to as a mass loss. Mass loss occurs because water artificially flows out of the mesh.



To minimize the abrupt change in flow direction, element edges can be curved by slightly moving the midside node. This can be done by hand using the Select Nodes tool with the nodes unlocked. Moving large numbers of nodes becomes tedious. However, element edges along a selected nodestring can be smoothed by SMS with the Smooth Nodestring command.

Normally, element edge smoothness is only a concern along the mesh boundary. However, if the analysis includes regions that become dry, interior boundaries should also be smoothed. To avoid smoothing corners that should be sharp, SMS provides a Smooth nodestring feature angle in the Element Options dialog. A corner will only be smoothed if it is less than the specified angle.

Related Topics

- 2D Mesh Elements Menu

Find Element

The Find Element command from the Elements Menu is used to locate an element either with a specific ID, or surrounding a specific location. When this command is executed the Find Element dialog opens.

When the Find by ID option is selected, then the element with the specified ID is highlighted in red. If there is no element with the specified ID, then an error message is given. Conversely, when the Find by nearest (x,y) coordinate option is selected, the element which surrounds to the specified (x, y) location is highlighted in red. With either of these methods, if the current tool is the Select Elements tool, then the found element becomes selected in addition to being highlighted.

Related Topics

- 2D Mesh Elements Menu

Mesh Element Options

Certain parameters governing the creation and manipulation of nodes are set using the Element Options dialog, which is opened by selecting the Options command from the Elements menu. This dialog is divided into four sections.

General Options

The General Options section of the Element Options dialog specifies the following parameters for general element operations:

Select thin triangle aspect ratio

When SMS finds thin elements, only elements with an aspect ratio (element width divided by element length) less than this value are selected. This value is also used by in the model checker mesh quality checks.

Merge triangle feature angle

This angle should be between zero and ninety degrees. Any two adjacent triangles are merged into a quadrilateral if all angles in the resulting quadrilateral are greater than the merge triangles feature angle.

Smooth nodestring feature angle

When a nodestring is smoothed, the smoothing will not be applied around a corner whose angle is greater than this value. See the convert elements article for a discussion on nodestring smoothing.

Preserve material boundaries

When turned on, triangles will not be merged into quadrilaterals if they are assigned different materials types, even if they satisfy the merge triangle feature angle criteria.

Materials

The Materials section of the Element Options dialog controls how materials are assigned to elements using the following options:

Set default material

This defines the default material assigned to elements as they are created.

Assign default material

When turned on, the material selected as the default material is assigned to selected elements when the assign material command is issued.

Prompt for material when assigning

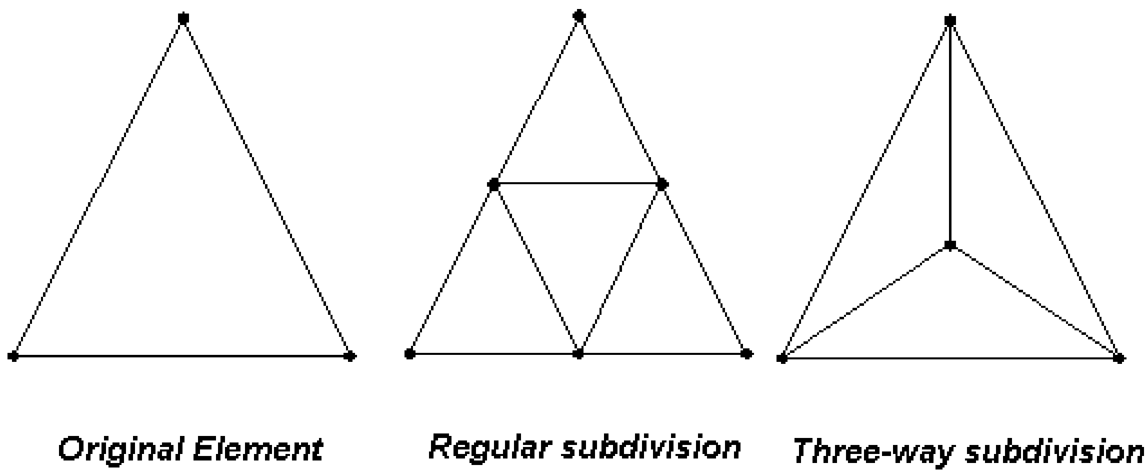
If this option is selected, the user must select from a list of existing materials to assign to the selected elements when the assign material command is issued.

Refine Elements

In some cases, a mesh does not have enough elements in a particular region of the mesh to ensure stability. Rather than inserting supplemental nodes and re-creating the mesh, it is possible to refine a selected region of the mesh using the Refine Elements command in the Mesh menu. This increases the mesh density of a selected area of the mesh. If no elements are selected, the entire mesh is refined. The elevations of the new nodes are interpolated from the existing nodes.

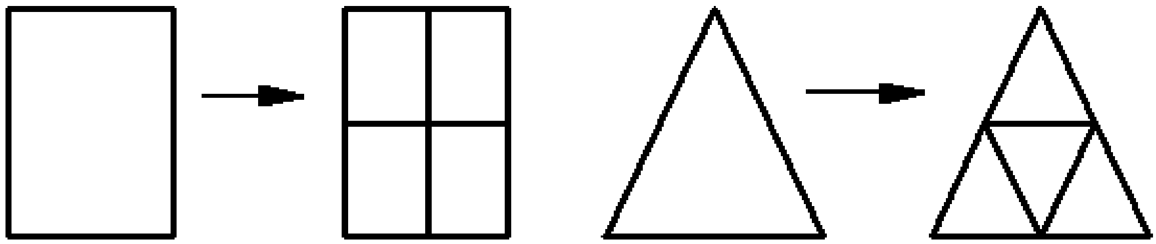
Three-way subdivision

Refines triangular element into three elements. Quadrilateral elements are not refined.



Regular subdivision

Refines triangular and quadrilateral elements into four elements.



Relax Elements

The Relax Elements section of the *Element Options* dialog controls the following relaxation parameters:

Number of iterations

This is the number of iterations to perform during the relaxation process.

Interpolate Z from existing mesh

When turned on, the nodal Z coordinate is interpolated from the old mesh so that the contours do not change. When this is turned off, the nodal Z coordinates are not changed when they are moved.

Lock nodes on nodestrings

Preserve material boundaries when relaxing. Previous versions of SMS would lock any nodes on a material or mesh boundary. Starting with version 7.0 of SMS, nodes on these boundaries will slide along the boundary unless it is part of a nodestring and this option is turned on.

Relaxation Method

- **Area relax** – equalize the area of elements adjacent to each node.
- **Scatter relax** – space the nodes according to the specified size dataset.
- **Angle relax** – equalize the angle of elements adjacent to each node.

Boundary Relax

Allow sliding on mesh boundary

When turned on, relaxation may modify the location of nodes on the mesh boundary.

Material boundaries

- **Allow relax** – Allows nodes located on material boundaries to move in all directions.
- **Preserve** – Prevents nodes located on material boundaries from moving.
- **Allow sliding** – Allows nodes located on material boundaries to move along the material boundary.
- **Sliding angle** – When a node is smoothed, the smoothing will not be applied around a material boundary whose angle is greater than this value.

Related Topics

- 2D Mesh Elements Menu
- Convert Elements

Relax Elements

The process of creating and editing a finite element mesh can result in poor quality elements. These elements may have poor interior angles or may violate the area change guideline for adjacent elements. The Relax command from the Elements menu can improve adjacent element areas and interior angles by moving nodes. This command moves nodes to improve the elements shape. Several options are available. Relaxation is an iterative process. The number of iterations performed and other options are specified in the Element Options dialog. If no elements are selected, then the relaxation is performed on all elements in the mesh.

Related Topics

- Element Options Dialog
- 2D Mesh Elements Menu

Fix Bad Area Transitions

Fix Bad Area Transitions (Elements Menu)

The process of creating and editing a finite element mesh can result in poor quality elements. These elements may violate the area change guideline for adjacent elements, specifically, the area of the smaller of the adjacent elements divided by the area of the large element may be less than a recommended ratio. This ratio is set in the Mesh Quality entry of the 2D Mesh Display Options. The Fix Bad Area Transitions command from the Elements menu can improve adjacent element areas by removing nodes. The adjacent elements of each node in the mesh are examined for guideline violations. If more that one are found, a calculation is made to determine if the removal of the node and the retriangulation this would cause would maintain compliance for the newly formed elements and their neighbors. If this is the case the node is removed.

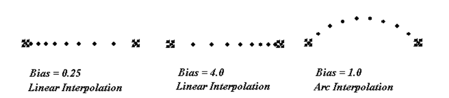
Related Topics

- 2D Mesh Elements Menu
- Mesh Module Display Options

3.8.b.3.2. 2D Mesh Nodes Menu

2D Mesh Nodes Menu

2D Mesh Nodes are the basic building blocks of elements in finite element meshes. Nodes are also required to create nodestrings and assign boundary conditions. The following commands are available when working with 2D Mesh Nodes:

Command	Description
Interpolation Options	The Interpolation Options item from the Nodes menu opens the Node Interpolation Options dialog. Using the options that are set in this dialog, a set of new nodes can be interpolated between any two selected nodes. If two nodes are selected when this dialog is invoked, the distance between the two nodes is displayed at the top of the dialog. The number of new nodes can be specified in three ways: Number of intervals in string. If this option is chosen, the number of new nodes is one less than the number of intervals specified. Number of interpolated nodes. If this option is chosen, the number of new nodes is exactly specified. Total number of nodes in string. If this option is chosen, the number of new nodes is two less than the number of nodes specified. The Bias factor controls the distribution spacing of the new nodes. This factor can be any number between 0.1 and 10.0. A smaller factor will make new nodes be closer to the first selected node while a larger factor will make new nodes be closer to the second selected node. For example, a bias of 2.0 makes the first new node spaced twice as far as the last new node. The Linear/Arc option controls the distribution shape of the new nodes. The Linear option causes all new nodes to be in a straight line while the Arc option causes all new nodes to form an arc. If the arc option is used, a Radius must also be specified. The arc will be created counter-clockwise from the first selected node to the second.  Interpolation option examples.
Interpolate	After the interpolation options are set up, nodes can be interpolated between any two selected nodes by choosing the Interpolate item from the Nodes menu. This operation may be performed multiple times with a single set of interpolation options by selecting any two nodes and invoking the command again. The elevation of each new node depends on the Insert nodes into triangulated mesh option in the Node Options dialog (see section 1.6.8). If this option is turned on and the new node is inside the finite element mesh, then the elevation is interpolated from the mesh. If this option is turned off or the new node is not inside the finite element mesh, the elevation is interpolated from the two selected nodes.
Find Node	When the Find Node command is executed, the Find 2D Mesh Node Dialog opens allowing the user to search for a specific 2D mesh node.
Select or Delete Duplicate Nodes	Duplicate nodes are either selected or deleted, according to the option defined in the Node Options dialog. The menu item shows either Select Duplicate nodes or Delete duplicate nodes based on the setting. Two nodes are considered to be duplicates if they are closer together than the Tolerance in the Node Options dialog. When deleting duplicate nodes, elements attached to deleted nodes will also be removed, unless the Merge adjacent elements when deleting option is turned on in the Node Options dialog.
Select Disjoint Nodes	Disjoint nodes can be found automatically and selected by choosing the Select Disjoint Nodes option from the Nodes menu. Disjoint nodes are nodes that are not connected to any elements. Before saving a simulation, it is important to make sure there are no disjoint nodes in the mesh.
Locked	The nodes in a mesh can be dragged with the mouse cursor if they are unlocked and the Select Nodes tool is selected. The Locked item in the Nodes menu toggles on and off the node locked status. If nodes are locked, a check mark is shown next to the menu text. The default status is locked so that nodes are not accidentally moved.

Reduce Nodal Connectivity	This command searches through the active mesh looking for wagon wheel nodes. When such a node is found, SMS reduces the nodal connectivity by inserting new node(s).
Renumber	The Renumber command from the Nodes menu is used to order the IDs of the nodes and elements to make numeric calculations more efficient. The goal is to make the matrix used in calculations as diagonal as possible by having related nodes numbered with indices as close as possible to each other. SMS now utilizes a Cuthill-McKee global renumbering scheme to update these indices. When a mesh is generated, it is resequenced, however, after performing hand edits the mesh should be sequenced again. Multiple invocations of the scheme may result in slightly different sequences.
Transform	The Transform command from the Nodes menu is used to move a group of selected nodes. If there are no selected nodes, the transformation will be applied to all nodes of in mesh. When this command is executed, the Nodes Transform dialog opens. In this dialog, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available: Scaling, translation, datum conversions, and rotations are supported. By default, the image will be framed after the transformation takes place. However, this can be turned off by using the Frame image after transformation option.
Options	Parameters governing the creation and manipulation of nodes are set using the Node Options dialog.
Interpolate Nodal Boundary Conditions	If two non-adjacent boundary nodes have been assigned boundary conditions, and the two nodes are selected, this command interpolates the boundary conditions to each of the boundary nodes between the two.

Related Topics

- Mesh Module Tools
- Mesh Module

2D Mesh Node Options Dialog

Parameters governing the creation and manipulation of nodes are set using the Node Options dialog, which is opened by selecting the menu Nodes | Options.

Individual Node Options

- **Insert nodes into triangulated mesh** – When a node is created inside the mesh boundary with the Create Mesh Nodes Tool, it can become part of the mesh. If this option is turned off, new nodes are not added to the mesh triangulation and remain disjoint. This option also applies to nodes created using the Interpolate command from the Nodes menu.
- **Retriangulate voids when deleting** – When deleting a node is deleted, all elements attached to the node are also deleted. The void in the mesh left by the deleted elements can be automatically filled by triangulating the surrounding nodes. If this option is turned off, then the void will remain.
- **Node Z value** – The z-value of a node created with the Create Mesh Nodes Tool is based on the chosen option:
  - Interpolate z-value from mesh – The Z coordinate is determined by interpolation from the existing mesh. If this option is turned off or the node is created outside the existing mesh boundary, the default Z coordinate is assigned.
  - Assign default z-value – The Z coordinate is assigned the default value.
  - Prompt for z-value – A dialog will prompt for the Z coordinate of each node after it is created.
  - Interpolate z-value from active scatter – The Z coordinate is determined by interpolation from the active scatter set. If the node is created outside the active scatter set boundary, the default Z coordinate is assigned.

Duplicate Node Options

- **Merge adjacent elements when deleting –** When a duplicate node is removed, the adjacent elements are merged.
- **Tolerance –** Tolerance for duplicate node selection and deletion. Also used by the automated mesh generation algorithms of SMS as a minimum node spacing. The tolerance should be specified in feet or meters. If using a Geographic Coordinate System, the tolerance is automatically converted by SMS to meters.
- **Select/Delete duplicate nodes –** The Nodes | [Select/Delete] Duplicate Nodes command is based on this selection.

Related Topics

- 2D Mesh Nodes Menu
- Mesh Module

Find 2D Mesh Node Dialog

The Find 2D mesh Node dialog is used to locate a node using the following methods:

Find by ID

When the Find by ID option is selected, then the node with the specified ID is highlighted with a red circle. If there is no node with the specified id, then an error message is given.

Find by Nearest (x,y) coordinate

When the Find by nearest (x,y) coordinate option is selected, the node closest to the specified (x, y) location is highlighted with a red circle.

With either of these methods, if the current tool is the Select Mesh Nodes tool, then the found node becomes selected in addition to being highlighted.

Related Topics

- Mesh Module Menus
- Mesh Module Tools

Renumber

Renumbering a mesh improves the computational efficiency (how fast a model produces a result) of a numeric mesh but should not affect the end results.

To renumber a mesh select the renumber command in the nodes menu.

Upon execution of this command, the nodes and elements are renumbered using a global renumbering process known as the Cuthill-McKee or Inverse Cuthill-McKee scheme. Other global resequencing methods may be added in future versions. The Cuthill-McKee method searches for a global optimum, but since are often multiple options with the same efficiency level (bandwidth), invoking the command multiple times usually results in different numbering patterns. Each time a mesh is generated, SMS invokes a renumbering command. When nodes are manually added/removed from a mesh, it should be renumbered.

It is important to realize that after renumbering the finite element mesh, any previous boundary condition file or solution file is no longer valid! They must be resaved with the new indices.

Front Width and Band Width

There are two measures of efficiency of a matrix. These include front width and band width. Both can be computed in multiple ways from the grid. The Cuthill-McKee scheme has its own method of computing band width and reports the band width before and after renumbering. In addition, SMS provides an estimate as to how large the front width and half band width may become when running the finite element solver. These estimates are shown in the Mesh Information dialog, which can be opened by performing the "File | Get Info" command while in the Mesh Module.

Due to the number of questions that are asked regarding this subject, this section will attempt to describe, in a broad sense, why renumbering is important.

The finite element solvers use an iterative, banded numerical solver to solve the governing differential equations. If the computer had to simultaneously solve the thousands of equations, it would run out of memory. This is why it uses a banded solver. The front width and the half-band width determine the size of the matrix which is used by the finite element solvers. A smaller front width and band width lead to a smaller required matrix. The front width and band width depend on the node and element numbering of the finite element mesh. To minimize the front width and band width, the mesh should be renumbered.

Renumbering Nodes and Elements

Meshes with gaps in numbering or with random numbering can lead to errors or inefficient solutions with many finite element solvers. The nodes and the elements can be renumbered simultaneously in an efficient manner by selecting the *Renumber* item from the Nodestrings menu. A node string must be selected before renumbering the mesh.

The selected node string is used to specify where the renumbering process begins. The "row" of elements and nodes adjacent to the string is numbered first. The elements and nodes adjacent to the first set of nodes and elements are numbered next, and so on until all of the nodes and elements have been renumbered.

The nodes and elements are renumbered in a sequence that can be envisioned as a "moving front" that passes through the mesh. Since the front proceeds from one set of elements to an adjacent set of elements, disjoint portions of the mesh will not be visited in the renumbering process. Unvisited nodes and elements are numbered arbitrarily.

Related Topics

- Nodestrings Menu

Reduce Nodal Connectivity

Many finite element engines (including ADCIRC) have limits on the number of elements that may be attached to a single node. When many elements attach to a single element, the node appears as a hub with many spokes radiating from it. Thus, it is termed a wagon wheel node.

When many elements are connected to a single node, each element has a smaller interior angle. This results in more severe deformations in numerical space. Ideal triangular elements have internal angles of 60 degrees. Ideal quadrilateral elements have internal angles of 90 degrees. For triangles, this would result in six elements connected (or constructed using) a single node. For quadrilaterals, at most four elements would converge at each node.

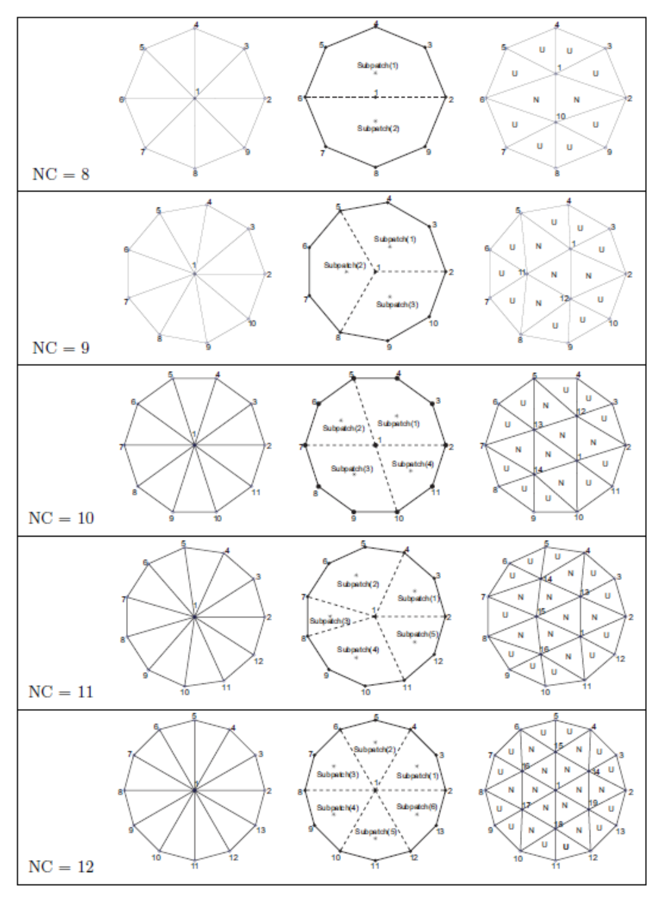
The node menu of the Mesh module includes the "Reduce Nodal Connectivity" command which inserts one or more new nodes in the area of wagon wheel nodes resulting in a maximum of 7 adjacent elements.

The command may need to be applied recursively. The first time may reduce connectivity from 12 or more to 8. Then a second application reduces connectivity to 6.

Specifically, if the current connectivity is:

- 8 elements – 1 new node added
- 9 elements – 2 new nodes added
- 10 elements – 3 new nodes added
- 11 elements – 4 new nodes added
- 12 elements – 6 new nodes added.

In each of these cases, the resulting elements all have connectivity of 6 elements. The patterns of insertion are illustrated below:



Related Topics

- Mesh Module Menus
- Mesh Module Tools

3.9 Particle Module

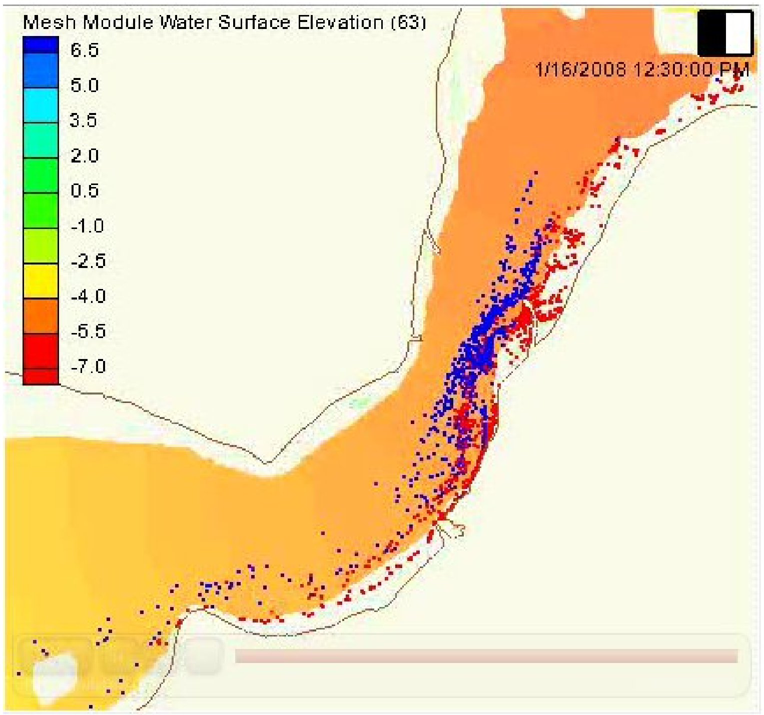
Particle Module

At a glance

- Visualize particle/path data
- Supports PTM module which computes particle positions through time based upon hydrodynamics and wave effects

The particle module contains tools used to work with particle data. Particles can have time varying location and scalar data. The particle module currently includes interfaces for:

- PTM – Lagrangian particle tracker designed to allow the user to simulate particle transport processes.



PTM Particle Trace

Particle Module Tools

See Particle Module Tools for more information.

Particle Module Menus

See Particle Module Menus for more information.

Particle Module Display Options

- Display Options

Related Topics

- SMS Modules
- Particle Tracking Model (PTM)

Particle Data Menu

Most of the SMS modules have a Data menu, but the items in this menu are different for each module. The Mesh Module commands include:

Dataset Tools

- Data Calculator
- Data Set Toolbox
- Create Data Sets
- Compute Grid Data Sets

Visualization Tools/Options

- Film Loop

Particle Module Display Options

The properties of all particle tracking data that SMS displays on the screen can be controlled through the Particle tab of the Display Options Dialog. This dialog is opened by selecting Display | Display Options from the menu bar, the display options macro, or the Ctrl+D quick keys.

The display options defined will only be applied to the active Particle Set displayed in the Project Explorer. The active Particle Set is listed at the top of this display options tab. To change to another set, close this window, left click on the desired Particle Set in the Project Explorer, and then reenter the display options.

The entities associated with the Particle module with display options are shown below. These entities also show an Options button to the right. For these entities, additional display controls are available. The available particle tracking display options are:

- **Particles**

  A symbol is placed around each particle. The user can specify the symbol, size, and color of these representations by using the Options button. The toggle controls the display of the particles. The Color based on sets whether the particles will be colored normally or dynamically. The choices include:

  - **Default** – Displayed using the color defined by the Options button.
  - **Active Dataset** – Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.
  - **Defined Dataset** – Displayed using the coloring of the specified dataset defined using the Select Set… button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a data set.

- **Particle tails**

  A tail is drawn out from the particle as it moves through time. The farther the particle has moved, the longer the tail may be; if the particle has remained stationary, then a tail may not appear. The user can specify the tail style, width, and color by using the Options button. The toggle controls the display of the particles. Tail Length, in seconds, controls the amount of tail displayed. The longer (time length) the particle has a tail, the longer the tail may be; if the particle has remained stationary for the duration of the tail length, then no tail will appear. Lengths can be fractions of time steps and the default is 10 time steps. Include symbol can be turned on to display little filled circles at time step positions within the tail. The Color based on sets whether the particle tails will be colored normally or dynamically. The choices include:

- **Default –** Displayed using the color defined by the Options button.
- **Active dataset –** Displayed using the coloring of the active dataset. Available only when a Particle Set exists with a dataset.
- **Defined data set –** Displayed using the coloring of the specified data set defined using the Select Set… button. The name of the specified set will be displayed beside this button. Available only when a Particle Set exists with a dataset.
- **Same as above –** Displayed using the coloring of the same dataset specified from the Particles. Available only when the Particles are colored based on a defined set and the set has been selected.

- **Particle path lines**

  A path line is drawn from the original position of a particle to every position the particle inhabits thereafter. The path line will remain even after the particle has settled or has crossed the edge of the domain. The user can specify the path line style, width, and color using the Options button.

All off unchecks all three particle display options (Particles, Particle tails, and Particle path lines). This also disables the Particle Display Filter since nothing is selected to be displayed.

All on checks all three particle display options.

Particle Display Filter allows the user to specify the range of Particles, Particle tails, and Particle path lines to be displayed. Filtering the particle set can increase the displaying speed and improve visibility of specific particles or groups of particles. The total Number of particles within the active particle set is displayed for convenience. Display every allows the user to display series of particles, i.e. an input of 1 displays every particle, but an input of 7 will display particles 1, 8, 15, 22, 29… and so forth. Begin with particle denotes the first particle within the range to be displayed. The input cannot be less than 1 or more than the displayed particle set total. End with particle denotes the last particle within the range to be considered for display. If the input is not a multiple of the Display every input plus one, then the particle will not be displayed. For example, an input of 46 with Display every input of 15 will display the last particle because 3 * 15 + 1 = 46. If the input was 45, the particles 1, 16, and 31 will only be displayed (particle 45 is considered, but is filtered out). The input cannot be less than the Begin with particle input or more than the displayed particle set total. One or more of the display options must be on to enable the filter controls.

Specific Dataset Color Options allows the user to Select a specific dataset and adjust its color options (similar to geometry contour options, but specific to each Particle Set data set) by clicking on Options. Color options is available only when a Particle Set exists with a dataset.

Related Topics

- Display Options
- Particle Module

Particle Module Menus

The following menus are available in the Particle Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data

Model Specific Menus

- PTM

Particle Module Tools

The Particle Module tools are contained in the Dynamic Tools portion of the tool palette when the Particle Module is active.

Select Particles

The Select Particles tool selects a single particle with a left mouse click. A group of particles can be selected by dragging a box around them. Particles may be added to the selection by holding the *SHIFT* key and selecting additional particles. The main use of selecting particles in the particle module is to query the particle properties. When a single particle is selected, the Info Window will show the ID and the edit window will show the location and value of the active particle data set. If exactly two particles are selected, the Info Window will also show the exact distance between the two selected particles. If more than one particle is selected, the minimum, maximum, and average value of the active particle data set will be shown.

Related Topics

- Particle Module

Particle Report

When creating a drogue plot, SMS can write out the following statistical information for each particle:

- The particle’s starting and ending location.
- The start and end times for each particle in decimals days.
- The total distance it traveled over the length of the animation.
- The minimum, maximum, and average velocity at which it traveled.

Be aware that when a particle leaves the domain, it can no longer be tracked so the ending location will be the point at which it left the domain. You can click the Browse button to set the name of the report file.

See Also:

Film Loop Setup – General Options

Film Loop Setup – Drogue Plot Options

Extract Particle Subset

This command writes a portion of a particle set to a new a particle file. This portion may be a subset of the timesteps for the particle solution, a subset of the particles, or both. You get to this command by right clicking on a particle set and choosing "Extract subset (times/particles)."

The required information for this command includes:

- The filename to write the particle subset to.
- The first and last timesteps that define the range of time you want export.
- Whether to write every timestep, every other timestep, every 3rd timestep, etc.

You may define a subset of the particles to export by setting up filters (see PTM Particle Filters). For example, you may decide to only write particles with a specific range of grain size, or particles from a specific source.

The extracted particle set will have the same datasets that exist in the original particle set.

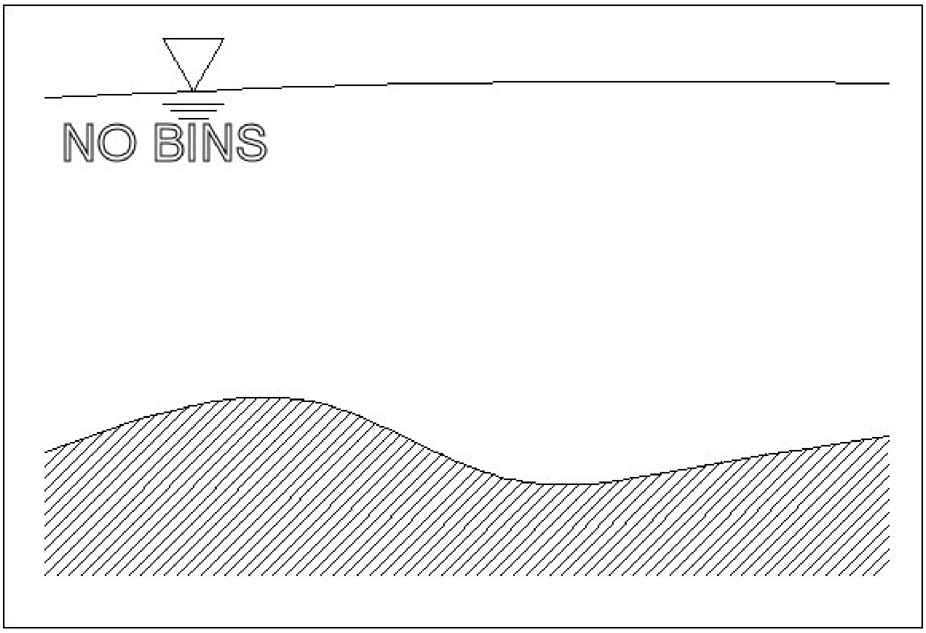
3.9.a. Particle Module Datasets

Particle Grid Dataset Bin Elevations

This article documents a feature that is under development

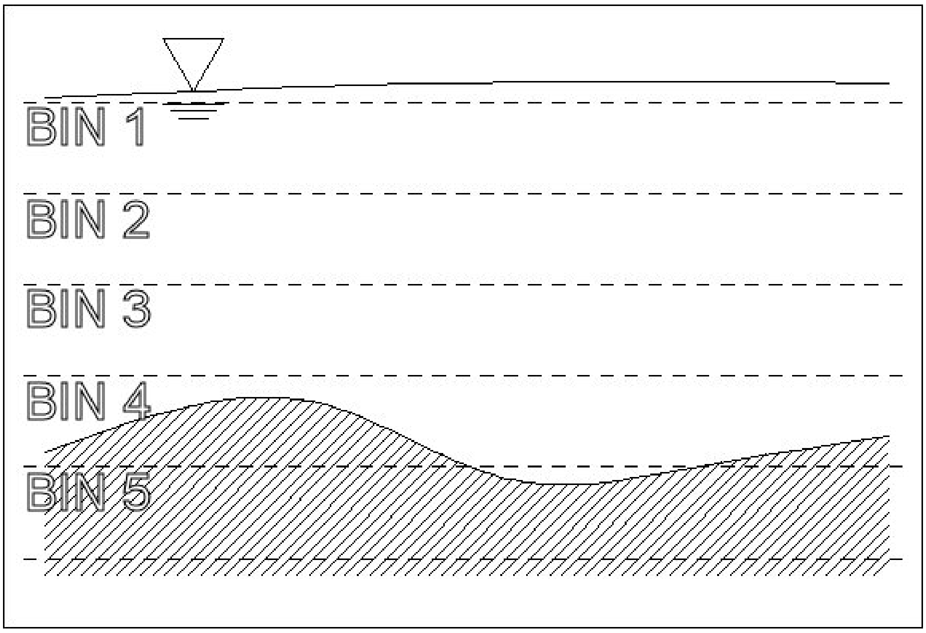
Datum Definition

The particle module compute grid dataset bin elevations dialog is accessed through the Compute grid datasets dialog. The bin elevations are specified according to the datum selected in the Compute grid datasets dialog.



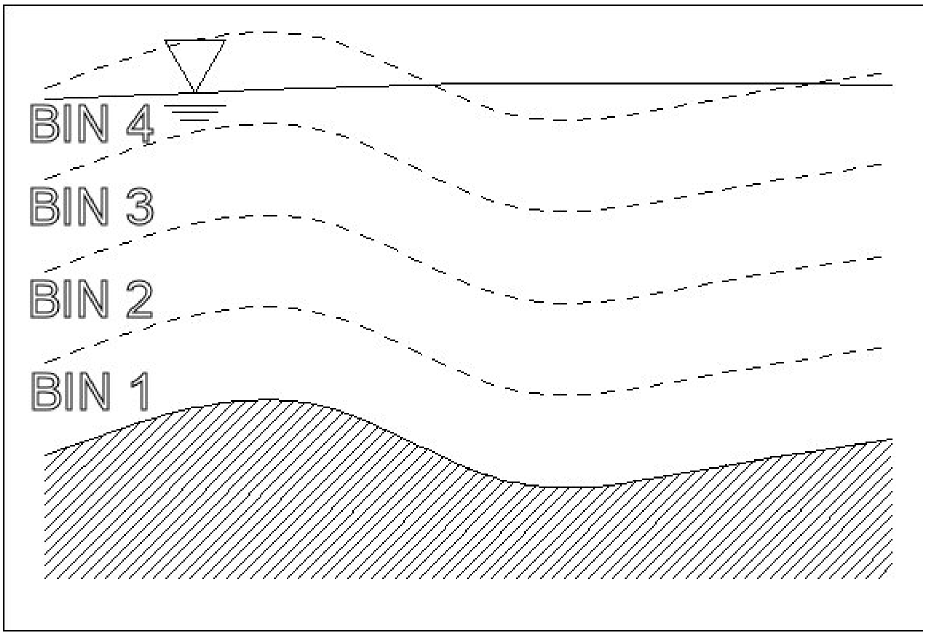
Fixed Datum

When using a fixed datum, the bins have a constant z-elevation. Be careful if your model uses a depth dataset. For example, if the water surface is at a constant value of 0.0 meters and the depth is a constant value of 10.0 meters, you would want your bins to range from 0.0 to -10.0 meters.



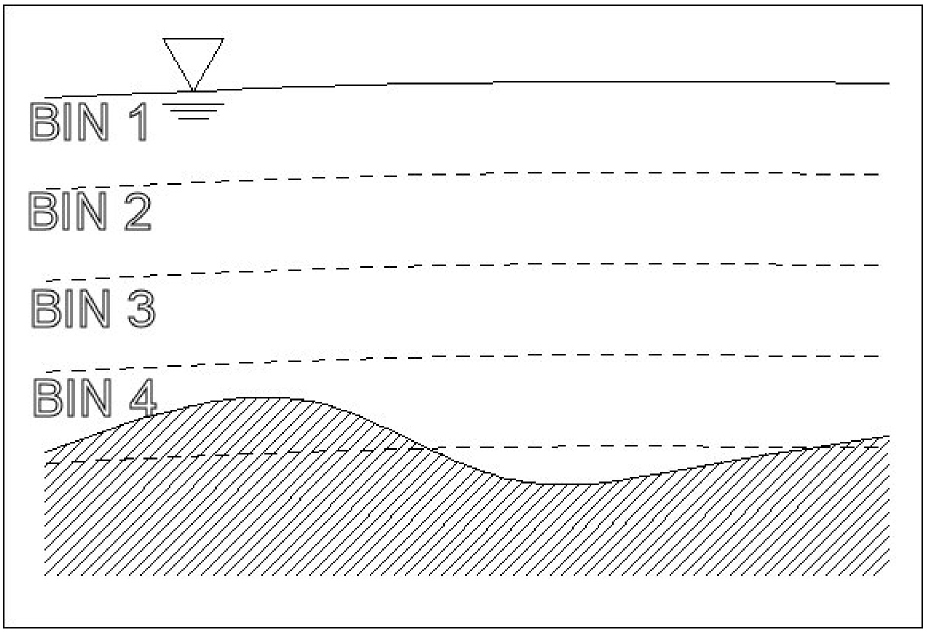
Bathymetry as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on bottom dwelling species, oyster beds, etc. Offsets should always be positive values.



Water Surface Elevation as Datum

When using the bathymetry as a datum, the bins are offsets from the specified bathymetry dataset. This is useful when determining the effects of concentrations on fish migrating at a specified depth, etc. Offsets should always be positive values.



Related Topics

- Particle Module Compute Grid Datasets Dialog

Particle Module Compute Grid Datasets

The particle module compute grid datasets dialog is accessed through the particle module data menu. The data sets are added to the active cartesian grid.

Currently the following grid datasets can be created in the particle module create datasets dialog:

- Particle Count
  - The number of particles in the Cartesian grid cell.
- Accumulation
  - The depth of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are inactive (based on the state dataset) and in the cell. The volume in each cell is divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting Cartesian grid dataset.
- Rate of accumulation
  - The change in accumulation (as described above) over time.
- Deposition
  - The change in depth of particles in the cartesian grid cell during the focus time. The volume of particles is calculated using the particle mass and density dataset for particles which have become inactive (based on the state dataset) during the focus time. This volume is then divided by the area of the cell to calculate an average depth in the cell. No voids ratio is included at this time, however the Data Calculator can be used to change the resulting cartesian grid dataset.
- Concentration
  - The concentration of particles in the Cartesian grid cell. The volume of particles is calculated using the particle mass and density dataset for particles which are active (based on the state dataset) and in the cell. This volume is then divided by the volume of the cell using the bathymetry and water surface elevation datasets.
- Exposure
  - The cumulative concentration relative to time in the Cartesian grid cell.
- Dosage
  - The exposure in the Cartesian grid cell during the focus time.

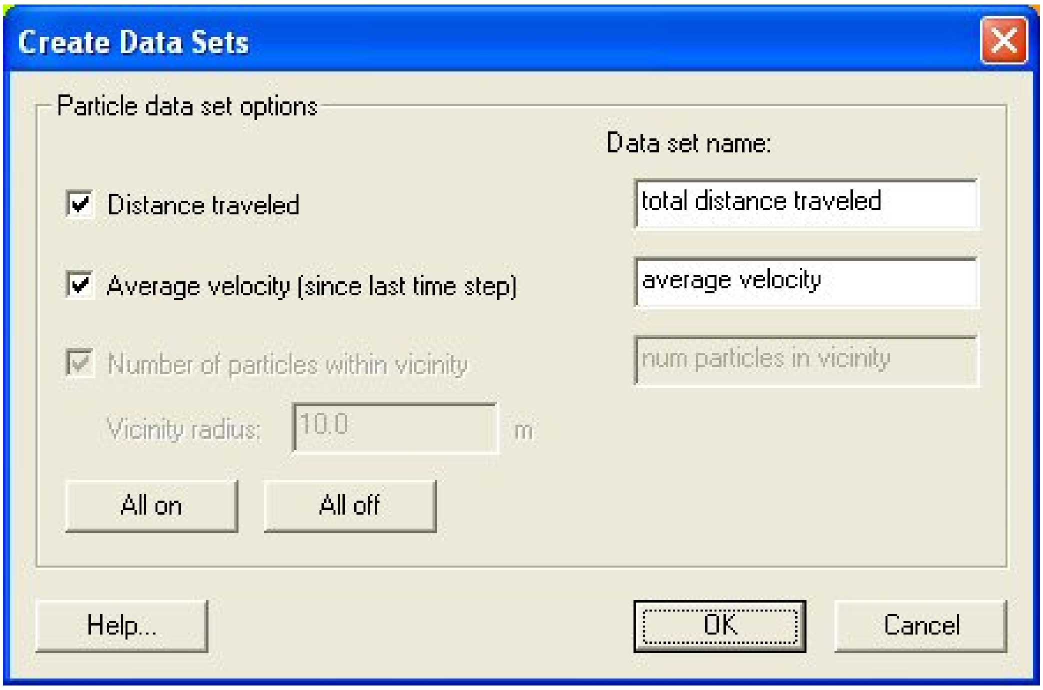
Related Topics

- Bin elevations dialog

Particle Module Create Datasets

The particle module create datasets dialog is accessed through the particle module data menu. Currently the following particle datasets can be created in the particle module create datasets dialog:

- Distance traveled
  - For each time step, computes the total distance each particle has traveled since the particle was born.
- Average velocity (since last time step)
  - For each time step, computes the average velocity of the particle since the previous time step.
- Number of particles within vicinity (under development)



PTM Create Grid Datasets – Fence Diagrams

The Particle Module Compute Grid Datasets page describes how to represent particle data on a rectilinear grid. The computations include things such as count, accumulation, and concentrations on the grid cells. In addition to computing these values on 2D grid cells, some of the datasets can be computed in layers creating 3D data. These datasets include concentration, exposure and dosage.

3D Fence Options

If the "create fence diagram" option is selected SMS will build a 3D mesh and datasets for each of the selected 3D datasets. SMS will also turn on the option to display fences (found in the display options dialog) and set the coverage used for fences. See 3D Fence Diagrams for information on adjusting the display of the fence.

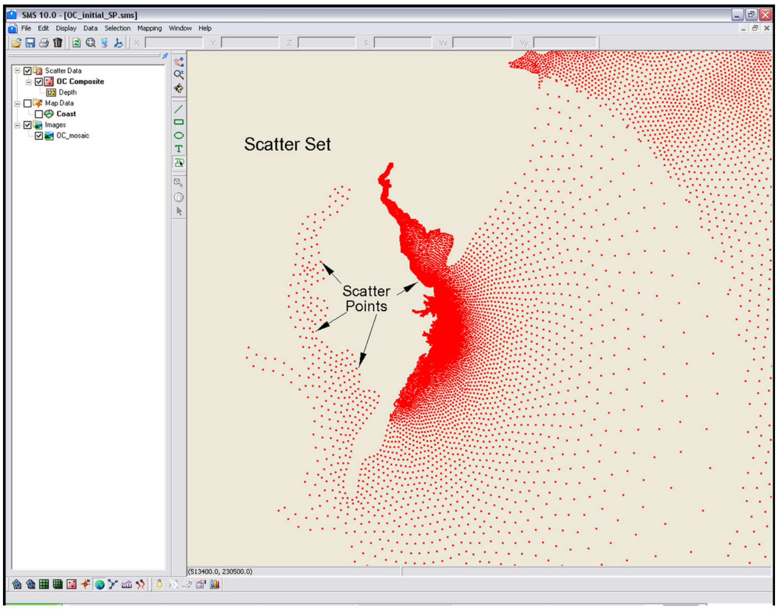
There will be two datasets generated for each type of 3D data computed. One dataset represents the concentration/exposure/dosage that is experienced by the cell. These values only make sense when applied to a volume. These datasets will display as a block filled value in each cell. Some people prefer viewing smooth contours rather than block filled values. The second dataset that is created represents the cell based values averaged to the nodes to provide for smooth contours. These datasets have "smoothed" in their names to distinguish them from the cell based data.

3.10 Scatter Module

Scatter Module

At a glance

- Used to create, edit, and visualize triangulated irregular networks
- DEMs can be read in and converted to TINs
- Filter scatter sets to eliminate redundant data
- Datasets can be interpolated to other modules (meshes, grids, etc)



Scatter Set Example

Overview

The Scatter Module (previously known as the Scattered Data Module) is used to interpolate spatial data values from groups of scattered data points or ordered grids (DEMs) to the other data types (i.e., meshes and grids). SMS supports three interpolation schemes including linear, natural neighbor and inverse distance weighted. The module is also used to view and edit survey data (i.e. SHOALS data).

Interpolation is useful for setting up input data for analysis codes. Generally, the data gathered from a site to be modeled varies in density. Generating a finite element mesh directly from these points would result in a very low quality mesh. Further this data does not lie in a grid for use as a finite difference grid. Interpolation allows the gathered data points to be used as background information. The user may then generate a base mesh or grid in the Mesh Module, the Grid Module or the Map Module. The only consideration of bathymetry for such a mesh or grid would be the definition of element edges along geometric or property features. The actual bathymetry comes from the scattered data. SMS interpolates this data to the created mesh or grid points.

Interpolation may also be used to create datasets for one mesh from data related to another mesh of the same region. For example, a user may have a mesh of a river reach for which analysis has been preformed. If a bridge is to be added to the reach, the mesh topology changes. The data from the first mesh can be converted to a scattered dataset and then interpolated to the second mesh. This data may be used as initial conditions for the second mesh, or compared to results of analysis run on the second mesh using the Dataset Toolbox.

A third purpose of interpolation is to create additional datasets from either observed, or calculated data.

Data Sources

There are various potential sources for background data in an SMS project. These include:

- **local surveys**

  Local surveys must be formatted into an SMS supported format. The most intuitive format and easiest to use is a tabular file of coordinates.

  If this data is to be augmented with previous models or historical surveys, the coordinate system of the local survey must be defined relative to the historical survey or a global coordinate system.

- **historical surveys**

  There are several sources of historic surveys. These include previous studies done by a modeler or company and compiled databases such as GEO-DAS or ETOPO. These data sources can be imported into SMS and used either as the basis of a finite computation domain (mesh or grid), or as a scattered dataset or DEM. Care must be taken into account the age and quality of the data and make sure all data sources are converted to a single coordinate system.

- **digital elevation maps**

  Digital Elevation Maps (DEMs) are regular structured grids of elevation values. Since the data is structured, it can be read, stored, displayed and utilized more efficiently than scattered datasets. These data sources are becoming more prevalent and can be obtained for topographic regions of the entire United States and several other area of the world from web sites such as Terraserver.

  Unfortunately, most DEMs available online do not include bathymetric portions of the domain, which makes their use in SMS limited. New data bases are being developed, but due to the dynamic nature of bathymetric information, the feasibility of an extensive database is very slight.

  It may be useful to convert scattered datasets into DEMs for faster processing inside of SMS.

- **electronic charts**

  Since surveys can be expensive to obtain, and DEMs may not be applicable, another option available for the hydraulic modeler is the use of topographic/bathymetric charts or historic nautical charts. If these types of maps can be digitized into an electronic format, they can be read into SMS and displayed on the screen. The goal is to create a scattered data set from this electronic chart. The steps to do this include:

  1. Scan the paper map and save it as an image (*.tif, *.jpg, ...).
  2. Register the image (you may want to mark the map with your register points prior to scanning it).
  3. Select the Create Vertex tool in the Data Module.
  4. Digitize (click on the image on the screen) to create a vertex on a contour line in the image.
  5. In the z edit box of the edit window set the z value to the contour value of the line.
  6. Digitize along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).
  7. Repeat steps 4–6 for each contour line. Spot elevations can be entered by setting the z value to the value of the spot elevation and then creating a vertex at that location.
  8. Triangulate the vertices once you are done.

  This method becomes tedious for larger areas, but is ideally suited for smaller areas where there are not too many contours to be digitized.

In addition, when DEM data is brought into SMS, the data is triangulated and stored as a scatter set.

It is also possible to convert CAD and GIS data into scatter sets. This is accomplished by right clicking on the object in the project explorer and selecting the "Map → Scatter" command. This command searches the data for triangular and quadrilateral faces and converts them to triangles in a triangulated surface (TIN). Points along contours or polylines are not converted using this command. In order to use these points in a scattered data set format, first convert them to feature objects in the Map module.

Practical Notes

Does SMS have a way of measuring the difference in volume between two bathymetric surveys?

To do this you need to do the following:

1. Interpolate the elevation from one survey onto the other.
2. Use the data calculator to compute the difference between the two elevations. You may want to do max(0.0, z1-z2) as well as max(0.0, z2-z1) to get both deposition and erosion volume.
3. Turn on the "Volume" option in the info options dialog.
4. Select the triangles you are interested in. The volume appears in the info window at the bottom of the screen. You can also direct these values to a file or another window through the Info Options settings.

How do I compare datasets from different scatter point sets?

Datasets within a scatter set are associated with the geometry of that scatter set. To compare datasets from different scatter sets, it is necessary to first interpolate the datasets to a common geometry. Below are guidelines on how to do this with a mesh and with a scatter grid.

- Mesh
    1. Interpolate the first dataset to mesh.
    2. Interpolate the second dataset to the mesh.
    3. Use the data calculator (*Data* | *Data Calculator*) to compare the two datasets.
- Scatter Grid
    1. Select first dataset.
    2. Select *Scatter* | *Interpolate to Scatter* | *…to Scatter Grid*. Specify extents and resolution of grid.
    3. Select second dataset.
    4. Select *Scatter* | *Interpolate to Scatter* | *…from other scatter set*. Specify the second scatter set.
    5. Select *Data* | *Data Calculator* with the new scatter grid selected to compare the two datasets.

Scatter Module Tools

See Scatter Module Tools for more information.

Scatter Module Menus

See Scatter Module Menus for more information.

Related Topics

- Digital Elevation Maps
- Mesh Generation
- Scatter Interpolation
- Scatter Module Display Options
- Scatter Sets

External Links

- Jun 2002 ERDC/CHL CHETN-IV-43 SHOALS Toolbox: Software to Support Visualization and Analysis of Large, High-Density Data Sets [1]

References

[1] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-43.pdf

Scatter Data Tools

The following tools are contained in the Dynamic Tools portion of the Tool Palette when the Scatter Module is active.

 Select Scatterpoints

The Select Scatterpoints tool is used to select scatter points (also known as vertices). A single point is selected by clicking on it. A second point can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple points can be selected at once by dragging a box around them. A selected point can be deselected by holding the *SHIFT* key as it is clicked.

Scatter points are locked so they are not accidently dragged. When a single point is selected, its location is shown in the Edit Window. The X, Y, and Z coordinates can be changed by typing in the edit field. For the Z coordinate, the active scalar function is updated when the Z coordinate changes. If multiple points are selected, the Z Coordinate field in the Edit Window becomes available. The value shown is the average scalar value of all selected points. If this value is changed, the new value will be assigned to all selected points.

With one point selected, the Edit Window shows the point id number. With two points selected, the Edit Window shows both point id numbers and the distance between the points. With multiple points selected, the Edit Window shows the number of selected points.

One or more scatter points can be deleted by pushing the **Delete** button or by going to **Edit | Delete**. Triangles attached to the deleted scatter points are deleted and the new void can be retriangulated.

 Select Triangle

The Select Triangle tool is used to select triangles. A single triangle is selected by clicking inside it. A second triangle can be added to the selection list by holding the *SHIFT* key while selecting it. Multiple triangles can be selected at once by dragging a box around them. Holding the *CTRL* key and dragging the mouse selects any triangles through which the line is drawn. A selected triangle can be deselected by holding the *SHIFT* key as it is clicked.

With one triangle selected, the Edit Window shows the triangle id number, its area, and its volume (area times average scalar value of the points connected to triangle). With multiple triangles selected, the Edit Window shows the number of selected triangles and their combined area and volume.

Create Triangle

Most triangles in SMS will be created using automatic triangulation. At times, however, it is necessary to create a triangle, especially after deleting triangles.

To create a single triangle:

Click on the corner points which will make the triangle, one-by-one. As each point is clicked, it becomes highlighted.

Alternatively, a box can be dragged around the points which will make the triangle. The triangle will not be formed if more than three points are selected. Before a new triangle is created, SMS checks if the new triangle will overlap other triangles. If this check fails, the new triangle is not created.

Swap Triangle Edge

The Swap Edges tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as avoiding an artificial dam in a channel.

Scatter Menu

The items in the Scatter menu in the Scatter module are described below. The menu items all work with the active scatter set unless otherwise noted.

General Commands

Listed below are the general commands in the Scatter menu.

Scatter Options

Scatter options are accessed through the *Scatter menu*, *Scatter Options* dialog in the Scatter Module.

Triangulation Options

This section lets the user adjust the maximum aspect ratio of a thin triangle. The aspect ratio is the ratio of the triangle width to the triangle height. All triangles with an aspect ratio less than what is specified are considered thin.

Long Triangles

This section contains options for deleting or selecting long triangles.

Individual Points

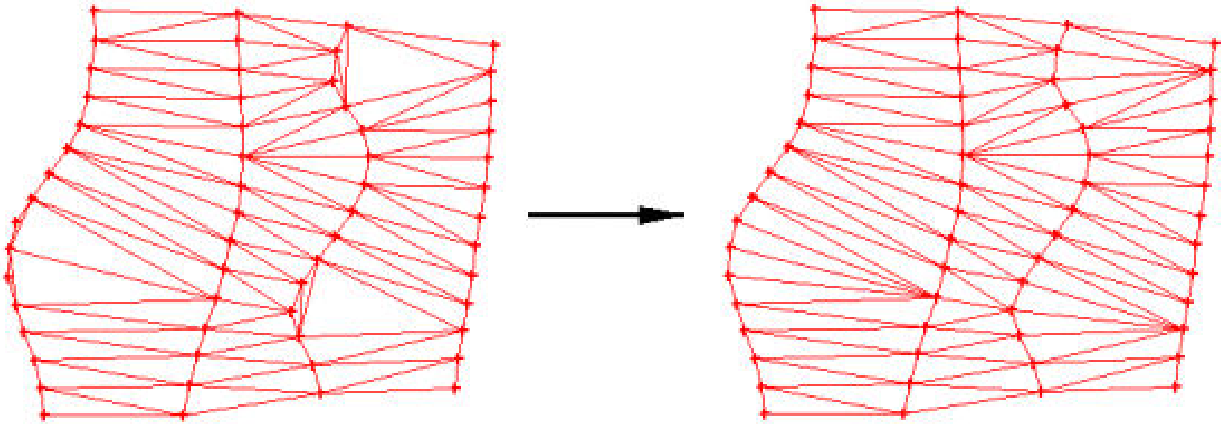
This section contains the retriangulate voids when deleting option. When scatter points are deleted, the triangles attached to the scatter points (if any) are also deleted. If this option is on, surrounding triangles are retriangulated to fill the void.

Triangulation Optimization Options

When the optimize triangulation command is invoked, the triangles are optimized in one of two ways:

- *Angle Optimization* – The triangles are swapped to conform to the Delaunay Criterion.
- *Area Optimization* (SHOALS) – The triangles are swapped to align with other triangles. The swapping is done by comparing the area of one triangle to its neighbor. The user sets a Bias. If the area of the smaller triangle is less

than the area of the larger triangle divided by the bias, the triangles are swapped. This is useful for optimizing the triangulation of surveys such as SHOALS surveys.



Duplicate Vertex Options

When the *Select/Delete Duplicate Points* menu item is selected, points within a tolerance of other points are selected or deleted. The user sets the Tolerance and whether to delete or select the points when the command is invoked.

Delete Scatter Set

This option is found in the Scatter module in the *Scatter menu*. If one scatter set exists, the user will be asked if they want to delete the active scatter set. If more than one scatter set exists, a dialog appears. The scatter sets can be flagged for deletion in this dialog. Double-clicking on a scatter set in the window or pushing the *Delete* button flags or unflags a scatter set for deletion. *Select All* or *Deselect All* will flag or unflag all sets. A set is flagged if the letter "d" appears to the left of the scatter name.

Merge Scatter Sets

Multiple scatter sets can be merged into a single scatter set using the *Merge Scatter Sets* dialog. The menu command *Scatter | Merge Sets* opens the *Merge Scatter Sets* dialog.

Selecting scatter sets to merge

The *Merge Scatter Sets* dialog contains a spreadsheet listing all of the scatter sets currently loaded into SMS. Scatter sets to merge are specified by checking the *Merge* box in the *Merge column* of the spreadsheet. When merging scatter sets, only one dataset is transferred to the merged scatter set. The dataset to be transferred is specified for each scatter set in the *Dataset column* of the spreadsheet.

The Priority column of the spreadsheet is only used if the Overlapping region option is set to *Delete lower priority scatter points*. This option is explained below.

Merge Options

The following options are available when merging scatter sets:

Merged scatter set options

- **Name** – Specify the name for the new, merged scatter set.
- **Delete original scatter sets** – The scatter sets to be merged are deleted after the new, merged set is created.

Overlapping region options

- **Merge all scatter points** – All scatter points from all scatter sets to be merged are combined into one set and retriangulated.
- **Delete lower priority scatter points** – In regions where scatter points and triangles overlap, the scatter points and triangles from the lower priority scatter set are deleted. The priority is based on the **Priority** column of the **Select scatter sets to merge** spreadsheet. The **Move up** and **Move down** buttons can be used to adjust the priority of the scatter sets when this option is selected.
  - **Maintain triangulation** – The triangulation of the original scatter sets is maintained. New triangles are created to connect the original scatter sets into a single, merged scatter set.

Merge Report

When the merge finishes a merge report will be displayed on the screen. This report shows statistics for the scatter sets that were merge such as number of vertices and triangles before and after the merge. If desired vertices are being deleted, check the duplicate points tolerance. This is found at Scatter→Scatter Options dialog.

Create Scatter Subset

All selected points from the original scatter set are moved from the original set into a new scatter set. The user is prompted for the name of the new scatter set. If all points for the current scatter set are selected, nothing occurs. The two scatter sets, the original and the new, are retriangulated.

Interpolate to Mesh

If mesh nodes exist, the Interpolation Dialog appears where the user sets the interpolation options. The scatter point datasets are then interpolated to the mesh nodes using the user specified interpolation options.

Interpolate to Cartesian Grid

If a cartesian grid exists, the Interpolation Dialog appears where the user sets the interpolation options. The scatter point function values are then interpolated to the center of each grid cell.

Interpolate to Scatter Grid

If a scattered dataset exists, the *Interpolate to Scatter Grid* menu item (Scatter module, Scatter menu) brings up the Grid Frame dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes OK, a new scatter set is created with scatter points at the corners of each grid cell. The original scatter set is interpolated to the new scatter grid set using linear interpolation.

Using the scatter grid is a form of data decimation: a dense scatter set can be represented as a less dense scatter set.

Interpolate to Nautical Grid

This option creates a nautical chart. A nautical chart divides a scatter set into bins and finds the maximum, minimum, and average depth over each bin. The Interpolate to *Nautical Grid* menu item (Scatter module, Scatter menu) brings up the *Grid Frame* dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes *OK*, a new scatter set is created with scatter points at the center of each grid cell. Three functions are created for each scatter point from the active scalar function of the original scatter set:

- **Average –** The average depth over each bin.
- **Minimum –** The minimum depth over each bin.
- **Maximum –** The maximum depth over each bin.

Requirements to Interpolate to a Nautical Grid

1. A scatter dataset must exist.
2. Active coverage type must allow grid frames to be created. The Cartesian Grid Module model coverage types allow the creation of grid frames.

Interpolate from Other Scatter

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. A dialog appears and the user selects the scatter set to interpolate from. That scatter set is interpolated to the active scatter set. The Options button brings up the Interpolation Options dialog, allowing the user to set the interpolation type. The interpolation uses an extrapolation value of 0.0.

Related Topics

- Scatter Module
- Scatter Module Menus

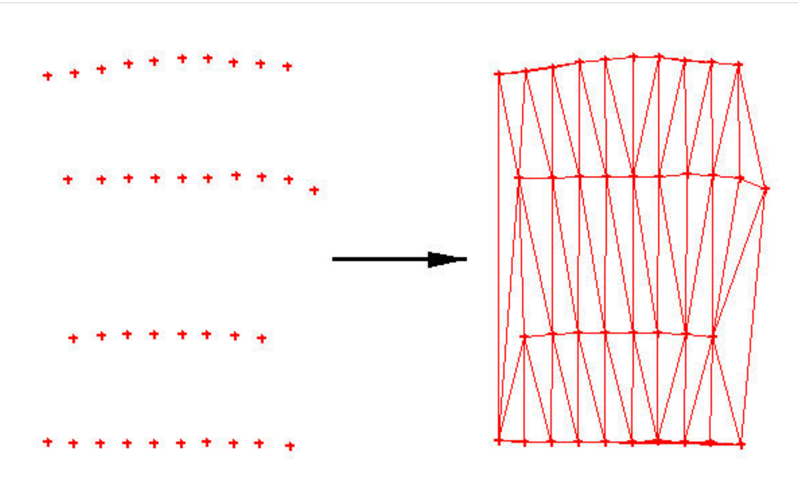
Scatter Triangles Menu

The items unique to the Scatter module are listed below. The menu items operate on the active scatter set unless otherwise noted:

General Commands

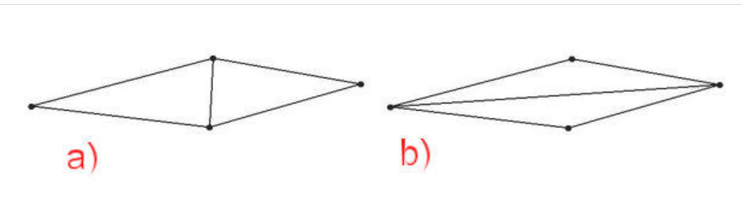
Triangulate

Scatter points or mesh nodes can be triangulated to form piecewise linear surfaces. For scattered data, these surfaces are also referred to as TINs (Triangular Irregular Networks). For mesh nodes, they form a finite element mesh. The points/nodes are connected into surfaces as scatter sets or meshes are created, but at times it may be necessary to reconnect the points (i.e. after deleting individual points/nodes or triangles/elements). New triangles are constructed in mass by triangulating a set of points when the Triangulate command from the Scatter Data menu is executed. The selected points are connected with a series of triangles. If points are not selected, then all points will be triangulated.



Delaunay Criterion

The resulting triangulation satisfies the Delauney criterion. The Delauney criterion ensures that no vertex lies within the interior of any of the circumcircles of the triangles in the network as shown below:

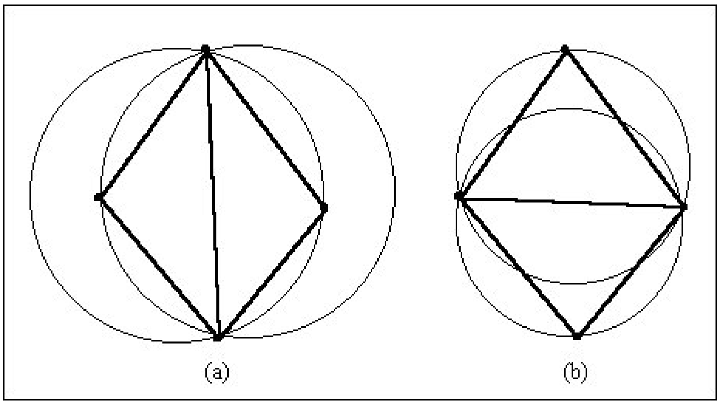


Two Adjacent Triangles Which (a) Violate and (b) Honor the Delauney Criterion.

The result of enforcing the Delauney criterion is that long thin triangles are avoided as much as possible.

Triangulate

The vertices associated with the active scatter set can be triangulated using the Triangulate command from the Triangles menu in the Scatter module. Mesh nodes (either the selected nodes, or all nodes) can be triangulated using the Triangulate command from the Elements menu in the 2D Mesh module.



Optimize Triangulation

At times, the user will perform manual mesh editing using the Swap Edge tool. This makes the Delauney criterion no longer hold. Selected elements can be returned to the Delauney state by choosing the Optimize Triangulation command from the Elements menu.

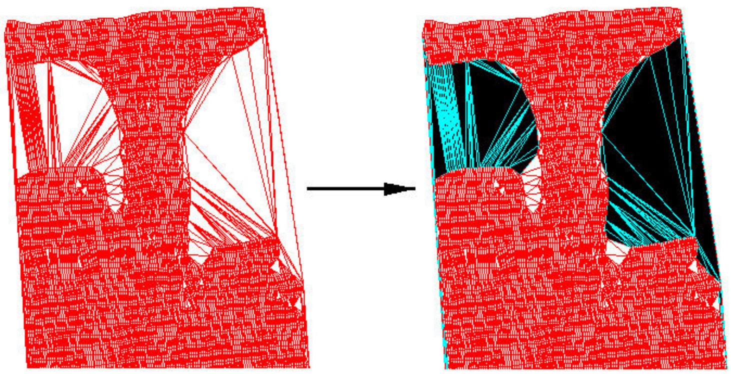
Select Thin Triangles

During the process of triangulation, a mesh of triangular elements is created around existing nodes. This usually creates triangular elements outside the desired mesh boundary. Many of these exterior triangles are very skinny, and some are virtually invisible. The Select Thin Triangles command from the Elements Menu finds and selects skinny triangular elements which are on the mesh boundary.

Thin triangles interior to the mesh will not be selected when this command is performed, since deletion of interior triangles would result in gaps in the mesh. After the thin triangles have been selected, they can be removed by selecting the Delete macro  .

Select/Delete Long Triangles

This option in the Scatter Module, Triangles Menu finds triangles longer than the length specified in the Scatter Options dialog. The Scatter Options also allows the user to select the option to delete or select the long triangles. Selecting/deleting long triangles is useful for deleting triangles that span regions where interpolation is not desired, such as over regions of land (see figure below, the selected triangles are over land).



Related Topics

- Scatter Module
- Scatter Module Menus

Scatter Vertices Menu

The menu items operate on the active scatter set unless otherwise noted:

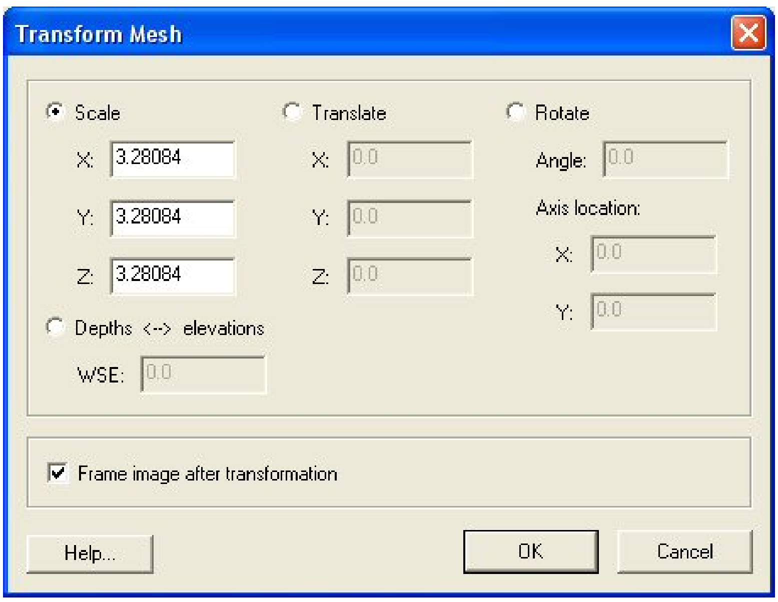
- **Locked** – The points in a scatter set can be dragged with the mouse cursor if they are unlocked and the Select Scatter Points tool is selected. The Locked item in the Vertices menu toggles on and off the locked status. If scatter points are locked, a check mark is shown next to the menu text. The default status is locked so scatter points are not accidentally moved.

Related Topics

- Scatter Module
- Scatter Module Menus

Transform

At a glance



Transform Data

- Data can be scaled, translated, rotated
- Depths/Elevations can be converted back and forth

The Transform command is used to move scatter points. The user is asked which will be transformed, the active set or all sets. In the dialog that appears, the transformation type can be chosen and then appropriate parameters can be entered. The following transformation types are available:

- **Scaling**: Scaling factors for the X, Y, and/or Z directions are entered. To prevent scaling a specific direction, the default value of 1.0 should be used.
- **Translation**: Translation values for the X, Y, and/or Z directions are entered. To prevent tranlation in a specific direction, the default value of 0.0 should be used.
- **Rotations**: When rotation is selected, the set of options on the right side of the dialog become available to define the center of rotation. If the *Specified Point* option is used, then the center of rotation is explicitly defined. Otherwise, after clicking the OK button from the *Nodes Transform* dialog, the user must click in the graphics window at the Point or on the Node about which the rotation should occur. The rotation will occur counter-clockwise by the specified angle around the specified center of rotation.
- **Datum Conversions**: Convert between elevation and depth data.

By default, the image will be framed after the transformation takes place. However, this can be turned off by using the *Frame Image After Transformation* option.

Related Topics

- Scatter Data Menu

3.10.a. Interface Components

Interface Components

The map module interface consists of the display options, menus, right click menus, and tools associated with the map module.

- Display Options
- Menus
- Right Click Menus
- Tools

Related Topics

- Display Options
- SMS Menus
- Dynamic Tools

Scatter Module Menus

The following menus are available in the the Scatter Module:

Standard Menus

See Menu Bar for more information.

Module Specific Menus

- Data
- Vertices
- Breaklines
- Triangles
- Scatter

Scatter Module Display Options

The properties of the scatter data SMS displays on the screen can be controlled through the Display Options dialog. The entities associated with the scatter module with display options are shown below. Some of these entities also show an Options button to the right. For these entities, additional display controls are available. The available display options include the following:

- **Points** – A symbol is drawn at each point. The user can specify the type, radius, and color of these symbols. The toggle below the Points item allows the user to specify that rather than coloring the symbols with the specified color, a contour color should be used based on the current scalar value at the point. This gives a contouring effect without generating/displaying the contours.
- **Triangles** – Triangle edges are drawn using the specified line attributes. Line attributes include color, thickness, and style (dashed/solid).
- **Boundary** – A line around the perimeter of the scatter set can be drawn. This is useful when the triangles are turned off. User specifies line color and thickness.
- **Contours** – The scatter contours are drawn for the active scalar data set for the active scatter set. All standard contour display options are supported for scatter contours.
- **Velocity Vectors** – The scatter vectors are drawn for the active vector data set of the active scatter set. Display options are set through the Vector Display Options dialog.
- **Inactive Color** – Only the active scatter set is displayed in its color. All other scatter sets are displayed using the inactive color. This helps to avoid clutter on the screen.
- **Nautical Grid**
- **Point Names** – The name of the selected scatter set can be changed.
- **Point Numbers** – The scatter point id number can be displayed next to each node. User selects font and color.
- **Scalar values** – The scalar value of the active function is displayed next to each point. The options button opens the Scalar Value Options dialog.
  - **Use contour color scheme** – Use the color ramp specified for the contours for text color rather than a specified color.

The display of individual scatter sets can be turned on or off through the Project Explorer.

Related Topics

- Display Options
- Scatter Module

3.10.a.1. Scatter Module Tools

Scatter Module Tools

The following tools are contained in the Dynamic Tools portion of the tool palette when the Scatter Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Scatter tool palette.

Tool	Tool Name	Description	Right Click Menu
	Select Scatterpoint	The Select Scatter Point tool is used to select scatter points (also known as vertices). A single point is selected by left-clicking directly on it. Multiple points can be selected at once by dragging a box. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. Additional scatter points can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new points without holding the *SHIFT* key will first clear the selection list and then add the newly selected points. A selected point can be removed from the selection list by holding the *SHIFT* key as it is reselected. Pressing the *ESC* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. Scatter points are locked by default so they are not accidentally dragged, but can be unlocked using the Scatter Vertices Menu. When a single point is selected, its location is shown in the Edit Window. The Z coordinates can be changed by typing in the edit field. The active scalar function is updated when the Z coordinate changes. If multiple points are selected, the Z Coordinate value shown is the average scalar value of all selected points. If this value is changed, the new value will be assigned to all selected points. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File* | *Info Options* command in the File Menu. Selected scatter points can be deleted by selecting the *Edit* | *Delete* menu command on the Edit Menu, by pressing the *DELETE* or *BACKSPACE* keys, or from the right-click menu. Triangles attached to the deleted scatter points are deleted. The resulting void can be retriangulated. This tool is available when one or more scatter points exist.	Process Boundary Triangles
	Create Scatter Point	The Create Scatter Point tool is used to place new scatter point in a Scatter Set. A single point or vertex is created at a time by left-clicking at the coordinate desired. The newly created point is selected to allow Z Coordinate changes in the Edit Window. Scatter points are locked so X and Y Coordinates can not be edited. This tool is always available, however, creating a scatter point is only allowed while in plan view.	N/A
	Select Breaklines	The Select Breaklines tool is used to select breaklines. Holding the *SHIFT* key while selecting breaklines will add breaklines to the selection that are not already selected and remove breaklines that are already selected from the selection. Multiple nodestrings can be selected by dragging a box around the breaklines. Holding the *CTRL* key and clicking will enable selection with a polygon. Holding the *CTRL* key and dragging will enable selection with an arrow. The Edit Window shows the number of breaklines selected.	N/A
	Create Breaklines	The Create Breaklines tool is used to create breaklines. Breaklinesare used to control the connectivity of a scatter set. To create a breakline:	N/A

		1. Click on a scatter vertex. The vertex will be highlighted in red and a prompt will be shown in the Help Window.
		2. Click on any vertex to add it to the breakline. The selected vertex is also highlighted in red and a solid red line is drawn between the two vertices. Continue adding vertices to the breakline in this manner.
		  1. Note: Vertices in the breakline can be adjacent, but this is not required. A breakline will usually be made of vertices which are not adjacent.
		  2. Press the *BACKSPACE* key to backup one vertex. Press the *ESC* key to abort the breakline creation.
		  3. Double-click a vertex or press the *ENTER* key to end the breakline creation.
		3. The *SHIFT* and *CTRL* keys assist in creating large breaklines which include sections made up of adjacent nodes. These can be used after at least one vertex has been selected and function as follows:
		  1. *SHIFT* – Holding down the *SHIFT* key and selecting another vertex will add to the breakline all vertices between the two. The path chosen is the shortest distance between the two vertices that follows triangle edges.
		  2. *CTRL* – Holding down the *CTRL* key and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going counter clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.
		  3. *CTRL* + *SHIFT* – Holding down both the *CTRL* and *SHIFT* keys and selecting another vertex will add to the breakline all vertices on the scatter set boundary between the two, going clockwise from the first vertex to the second vertex. Both vertices must be on the boundary of the scatter set or SMS will beep.	

	Select Triangle	The Select Triangle tool is used to select triangles. A single triangle is selected by clicking inside it. Multiple triangles can be selected at once by dragging a box or an arrow. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. To drag a selection arrow, left-click while holding the *CTRL* key (the *CTRL* key can be released after the click) and hold the mouse button and drag the mouse to form an arrow of the desired length and direction; release the button to impale and select the triangles through which the arrow passes. Additional triangles can be appended to the selection list by holding the *SHIFT* key while selecting by any method. Selecting new triangles without holding the Shift key will first clear the selection list and then add the newly selected triangles. A selected triangle can be removed from the selection list by holding the Shift key as it is reselected. Pressing the *ESCAPE* key will clear the entire selection list. Right-clicking will open a menu specific to this tool. When a single triangle is selected, its centroid location is shown in the Edit Window. If multiple points are selected, the Z value shown is the average scalar value of all selected triangles. The Graphics Window's status bar will display information on the selected items depending on the settings find through the *File* | *Info Options* command in the File Menu. Selected triangles points can be deleted by selecting the *Edit* | *Delete* menu command on the Edit Menu, by pressing the *Delete* or *BACKSPACE* keys, or from the right-click menu. The Scatter vertices (points) of the deleted triangles are not deleted. The resulting void can be retriangulated. This tool is available when one or more scatter triangles exist.	N/A
	Create Triangle	Most triangles in SMS will be created using automatic triangulation. At times, however, it is necessary to create a triangle, especially after deleting triangles. To create a single triangle, select the vertices of the desired triangle by left-clicking on each scatter point or by dragging a selection box. As points are selected individually, they will be highlighted. Also, triangle edge will be drawn between the first and second vertex. To remove the last highlighted point from the desired triangle, press the *DELETE* or *BACKSPACE* keys. To abort the creation of a triangle, press the Escape key. When three points are selected, SMS will try to create the desired triangle. To drag a selection box, left-click and hold the button while dragging the mouse to the appropriate dimensions; release the button to enclose and select the contents. The selection box must only contain the three desired vertices to create the triangle. If the new triangle will overlap existing triangles the triangle will not be created. If more then three points are selected (via a selection box) then SMS will not attempt to create a new triangle. This tool is available when three or more scatter points exist.	N/A

	Swap Triangle Edge	The Swap Edges tool is used to manually swap the edges of two adjacent triangles. This is useful in such cases as avoiding an artificial dam in a channel. Left-click on the desired triangle edge to swap it. The edges of the triangles will only be swapped if the resulting triangles are valid. This tool is available when two or more scatter triangles exist. Swapping edges is only allowed while in plan view.	N/A
	Create Mesh NodesContour	The Swap Edges tool is used to temporarily display a Z value on a TIN. Left-click at a location to display the value. The labels will not reappear if the graphic window view changes. This tool is available when at least one scatter triangle exists. Contour labeling is only allowed while in plan view.	N/A

Related Topics

- Scatter Module

Scatter Module Tools Right Click Menus

The following tools are contained in the Dynamic Tools portion of the tool palette when the Scatter Module is active. Only one tool is active at any given time. The action that takes place when the user clicks in the Graphics Window depends on the current tool. The following table describes the tools in the Scatter tool palette.

Tool	Tool Name	Right Click Menu
	Select Scatterpoint	• Delete – Deletes selected Scatterpoint(s) • Split Breaklines – • Assign Point Name – User defines point name.
	Select Breaklines	• Delete Selected – Deletes selected breakline • Merge Selected – Merges two or more selected breaklines together • Force Breaklines –
	Select Triangle	• Select Thin Triangles • Delete Long Triangles • Delete

Related Topics

- Scatter Module

3.10.a.2. Project Explorer Items

Project Explorer Items

In the Project Explorer, the Scatter Data folder houses all of scatter data information that is manipulated by the Scatter module. The Scatter Data folder will not appear in the Project Explorer until scatter data is opened in SMS. Once scatter data is opened, the Scatter Data folder will appear with the Scatter set item below it. There will also be elevation information that appears under the Scatter set item. The Scatter module may be activated by clicking on the Scatter set item in the project explorer. Once active, the Scatter module tool bar menu will appear to the right of the Project Explorer.

Related Topics

- Scatter Module Right Click Menus

Scatter Module Right Click Menus

The following Project Explorer right mouse click menus are available when the right mouse click is performed on a Scatter Module item.

Scatter Module Root Folder Right Click Menus

Right clicking on the Scatter module root folder in the project explorer invokes an options menu with the following options:

- **New Scatter Set –** Creates a new, empty scatter set.
- Display Options

Scatter Set Item Right Click Menus

Right clicking on a Scatter Set item in the Project Explorer invokes an options menu with the following module specific options:

- **Split –** Creates a new scatter set containing the selected scatter vertices. Selected scatter vertices are removed from the original scatter set.
- **Autogenerate breaklines –** Automatically creates breaklines following specified elevations.

Related Topics

- Project Explorer Right Click Menus
- Menu Bar

3.10.b. Functionalities

Scalar Value Options

The Scalar Value Options dialog controls the display options for the scalar values displayed next to each scatter point. The option to display the scalar value of the active scatter function next to each scatter point is turned on and off in the Scatter Module Display Options.

The following options are available in the Scalar Value Options dialog:

- **Decimal Precision –** The decimal precision used.
- **Alignment –** The location used. The following options are available:
  - On Point
  - Left
  - Center
  - Right
- **Text Options –** Change the color, size, and font for the text.
- **Location Options –** Display the text at:
  - Each point
  - On a grid (works well for dense data). The user specifies the x and y pixel spacing. Because the spacing is based on pixels, the spacing stays constant during zooming.

Related Topics

- Scatter Module Display Options
- Display Options
- Scatter Module

Data Module Commands

The items in the Scatter menu in the Scatter module are described below. The menu items all work with the active scatter set unless otherwise noted.

Delete Scatter Set

This option is found in the Scatter module in the Scatter menu. If one scatter set exists, the user will be asked if they want to delete the active scatter set. If more than one scatter set exists, a dialog appears. The scatter sets can be flagged for deletion in this dialog. Double-clicking on a scatter set in the window or pushing the *Delete* button flags or unflags a scatter set for deletion. *Select All* or *Deselect All* will flag or unflag all sets. A set is flagged if the letter "d" appears to the left of the scatter name.

Merge Scatter Sets

Multiple scatter sets can be merged into a single scatter set using the Merge Scatter Sets dialog. The menu command *Scatter* | *Merge Sets* opens the Merge Scatter Sets dialog.

Selecting scatter sets to merge

The *Merge Scatter Sets* dialog contains a spreadsheet listing all of the scatter sets currently loaded into SMS. Scatter sets to merge are specified by checking the *Merge* box in the *Merge* column of the spreadsheet. When merging scatter sets, only one dataset is transferred to the merged scatter set. The dataset to be transferred is specified for each scatter set in the *Dataset* column of the spreadsheet.

The Priority column of the spreadsheet is only used if the Overlapping region option is set to *Delete lower priority scatter points*. This option is explained below.

Merge Options

The following options are available when merging scatter sets:

Merged scatter set options

- **Name** – Specify the name for the new, merged scatter set.
- **Delete original scatter sets** – The scatter sets to be merged are deleted after the new, merged set is created.

Overlapping region options

- **Merge all scatter points** – All scatter points from all scatter sets to be merged are combined into one set and retriangulated.
- **Delete lower priority scatter points** – In regions where scatter points and triangles overlap, the scatter points and triangles from the lower priority scatter set are deleted. The priority is based on the **Priority** column of the **Select scatter sets to merge** spreadsheet. The **Move up** and **Move down** buttons can be used to adjust the priority of the scatter sets when this option is selected.
    - **Maintain triangulation** – The triangulation of the original scatter sets is maintained. New triangles are created to connect the original scatter sets into a single, merged scatter set.

Merge Report

When the merge finishes a merge report will be displayed on the screen. This report shows statistics for the scatter sets that were merge such as number of vertices and triangles before and after the merge. If desired vertices are being deleted, check the duplicate points tolerance. This is found at Scatter→Scatter Options dialog.

Create Scatter Subset

All selected points from the original scatter set are moved from the original set into a new scatter set. The user is prompted for the name of the new scatter set. If all points for the current scatter set are selected, nothing occurs. The two scatter sets, the original and the new, are re-triangulated.

Interpolate to Mesh

If mesh nodes exist, the Interpolation Dialog appears where the user sets the interpolation options. The scatter point datasets are then interpolated to the mesh nodes using the user specified interpolation options.

Interpolate to Cartesian Grid

If a cartesian grid exists, the Interpolation Dialog appears where the user sets the interpolation options. The scatter point function values are then interpolated to the center of each grid cell.

Interpolate to Scatter Grid

If a scattered dataset exists, the Interpolate to Scatter Grid menu item (Scatter module, Scatter menu) brings up the Grid Frame dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes OK, a new scatter set is created with scatter points at the corners of each grid cell. The original scatter set is interpolated to the new scatter grid set using linear interpolation.

Using the scatter grid is a form of data decimation: a dense scatter set can be represented as a less dense scatter set.

Interpolate to Nautical Grid]

This option creates a nautical chart. A nautical chart divides a scatter set into bins and finds the maximum, minimum, and average depth over each bin. The Interpolate to Nautical Grid menu item (Scatter module, Scatter menu) brings up the Grid Frame dialog. The user positions the purple grid frame and sets up the number of rows and columns in the grid. When the user pushes OK, a new scatter set is created with scatter points at the center of each grid cell. Three functions are created for each scatter point from the active scalar function of the original scatter set:

- **Average –** The average depth over each bin.
- **Minimum –** The minimum depth over each bin.
- **Maximum –** The maximum depth over each bin.

Requirements to Interpolate to a Nautical Grid

1. A scatter dataset must exist.
2. Active coverage type must allow grid frames to be created. The Cartesian Grid Module model coverage types allow the creation of grid frames.

Interpolate from Other Scatter

This option interpolates one scattered data set to another set. Two sets must exist for this option to be enabled. A dialog appears and the user selects the scatter set to interpolate from. That scatter set is interpolated to the active scatter set. The Options button brings up the Interpolation Options dialog, allowing the user to set the interpolation type. The interpolation uses an extrapolation value of 0.0.

- Interpolate to Map

This option allows the user the convert Scatter Breaklines to Featuer Arcs. Right-click on Scatter Module in the Project Explorer to convert.

Related Topic

- Scatter Module
- Scatter Data Menu
- Scatter Triangles Menu
- Grid Frame Dialog

Scatter Datasets

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered datasets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered dataset to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the Scatter Set combo box in the top Edit Window. Whenever a new scatter set is created, it becomes the active set.

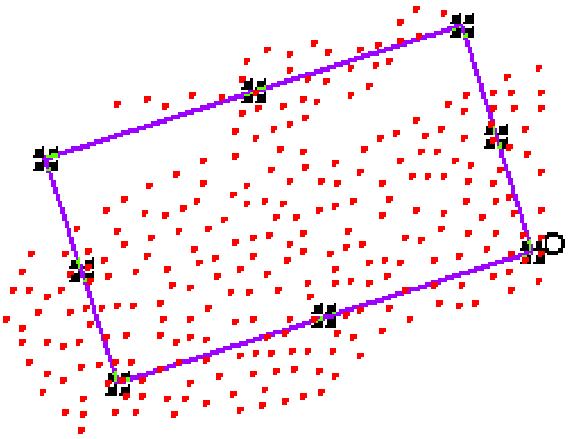
Related Topics

- Scatter Module

Grid Frame Dialog

Grid Frames are created from:

- Feature Objects | Grid Frame in the Map module for Cartesian Grid coverages
- Scatter | Interp to Scatter Grid in the Scatter module
- Scatter | Interp to Nautical Grid in the Scatter module



The grid frame is interactive with the Graphics Window and the Grid Frame dialog. Interactive options include:

- **Move Frame** – Click inside or on an edge where the frame is not highlighted and drag.
- **Resize Frame** – Click a highlighted corner or edge and drag to resize.
- **Rotate Frame** – Click inside the circle near the bottom right corner of the frame and drag to rotate the frame.
- – Redraw the screen.
- – Zoom to the extents of the data in the screen.

Dialog options include:

- **Grid Origin** – Set the bottom left corner location of the grid frame.
- **Angle of rotation** – Set the angle the grid frame will be rotated counter-clockwise from the +x axis.
- **Snap to grid** – After turning this option on, push the button to see grid snap points overlay the screen. The Snap Spacing controls the spacing of the grid snap points. As the user drags and resizes the grid frame, the grid frame edges snap to the grid snap points.
- **New/Delete Grid** – Available for Cartesian Grid frame only. Create or delete the current grid frame. For the other Scatter interpolation applications, a temporary grid frame is created for the specific application and then deleted.
- **Grid Dimensions** – Set the length and width of the grid frame. The Cell Dimensions are updated: Cell X = Grid X / Number of Columns, Cell Y = Grid Y / Number of Rows.
- **Cell Dimensions** – Set the length and width of each cell for the grid to be created.. The Grid Dimensions are updated: Grid X = Cell X * Number of Columns, Grid Y = Cell Y * Number of Rows.
- **Number of Columns/Rows** – Set the number of rows and columns for the grid to be created.
- **Options** – Set the interpolation options for the Scatter applications. Not used for the Cartesian Grid application.

Related Topics

- Feature Objects Menu

Use of DEMs in the Scatter Module

The Scatter Point Module is used to visualize and apply various types of data. This data typically comes from surveys, digital maps, previous numerical analysis or digitization on screen. The data is stored as sets or groups of 2D scattered data points with associated values. The most common value is bathymetry and is used to create the geometric representation of the area being modeled.

SMS connects the scattered data points into triangles forming a Triangulated Irregular Network (TIN). TINs can be contoured, displayed in oblique view with mapped images and hidden surfaces removed, and several other display options that can be set to visualize and understand the terrain surface better. TINs are used for a source of bathymetric or other data in a numerical model. TINs can also be used to compute areas, volume, distances, gradients and several other geometric parameters.

SMS applies data from scattered datasets to finite element networks or grids via interpolation. This allows poorly distributed elevation data to be assigned to a well-structured set of elements to create the bathymetry of the entire mesh. A variety of interpolation schemes are supported. Internally, the scattered data sets are triangulated to create surfaces for continuous interpolation. Since the connectivity of the triangulation affects the interpolation, SMS provides tools to allow for the manipulation of this triangulation. The triangulation also allows contouring of the scattered data set to visualize the data.

Multiple scatter point sets can exist at one time in memory. One of the scatter sets is always designated as the "active" scatter point set. The active scatter set can be changed by changing the Scatter Set combo box in the top *Edit Window*. Whenever a new scatter set is created, it becomes the active set.

Related Topics:

- Data Acquisition
- TINs

Scatter Options

Scatter options are accessed through the Scatter menu, Scatter Options dialog in the Scatter Module.

Triangulation Options

This section lets the user adjust the maximum aspect ratio of a thin triangle. The aspect ratio is the ratio of the triangle width to the triangle height. All triangles with an aspect ratio less than what is specified are considered thin.

Long Triangles

This section contains options for deleting or selecting long triangles.

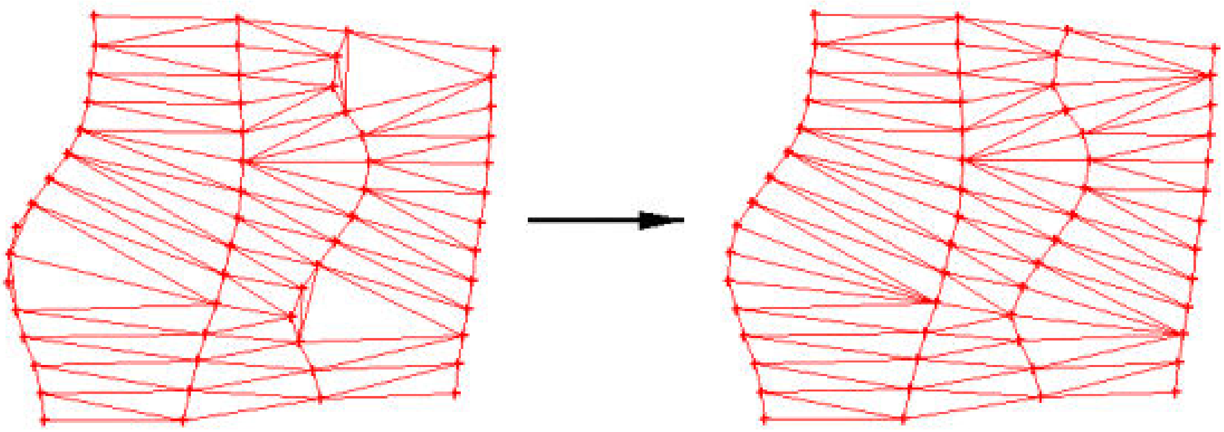
Individual Points

This section contains the Retriangulate voids when deleting option. When scatter points are deleted, the triangles attached to the scatter points (if any) are also deleted. If this option is on, surrounding triangles are retriangulated to fill the void.

Triangulation Optimization Options

When the optimize triangulation command is invoked, the triangles are optimized in one of two ways:

- Angle Optimization – The triangles are swapped to conform to the Delaunay Criterion.
- Area Optimization (SHOALS) – The triangles are swapped to align with other triangles. The swapping is done by comparing the area of one triangle to its neighbor. The user sets a Bias. If the area of the smaller triangle is less than the area of the larger triangle divided by the bias, the triangles are swapped. This is useful for optimizing the triangulation of surveys such as SHOALS surveys.



Duplicate Vertex Options

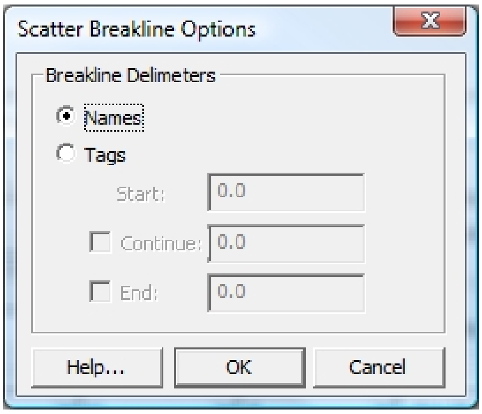
When the Select/Delete Duplicate Points menu item is selected, points within a tolerance of other points are selected or deleted. The user sets the Tolerance and whether to delete or select the points when the command is invoked.

Related Topics

- Scatter Menu

Scatter Breakline Options

When importing a Scatter Set using the File Import Wizard, selecting "Breaklines" as a Field will open the Scatter Breakline Options dialog. Breaklines are useful for maintaining the correct triangulation in a TIN.



Scatter Breakline Options

Breakline Delimeters

The available breakline delimeter options and samples of the associated file formats are:

- **Names –** Breaklines are identified using a unique name or ID
- **Tags –** Breaklines are identified using "start" and optionally "continue" and "end" values

No vertex can be in more than one breakline, and the breakline must be defined sequentially in the data file.

Named Breaklines

The breakline column of the data file includes the name of the breakline this vertex belongs to. If the breakline column is empty, the associated vertex is not included in any breaklines.

Example Files

Example of a tab delimited file using breakline names:

```
xcoord      ycoord     zcoord   name
215962.9    85203.098  1.483    Breakline1
215957.638  85193.069  1.483    Breakline1
215963.278  85184.35   1.483    Breakline1
215979.111  85179.328  1.483    Breakline1
216056.51   85209.371  1.483    Breakline1
215992.462  85201.477  7.034    Breakline2
216127.386  85264.681  7.034    Breakline2
216267.187  85327.936  7.034    Breakline2
216371.217  85381.431  7.034    Breakline2
219261.939  90247.944  8.763
219461.211  90220.556  9.167
219678.994  90179.064  9.468
```

Tagged Breaklines

The breakline column of the data file includes a tag or string defining when a breakline starts and stops. There are three types of tags including:

- **Start** – Identifies the start of a new breakline
- **Continue** – Indicates the vertex should be included in the current breakline
- **End** – Identifies the end of the current breakline

These tags may be used in a number of configurations:

- **Start**, **Continue**, **End** – When all three tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter should have a **Continue** tag until the line with the **End** tag. Lines with no entry in the Breakline column between breaklines are read as vertices not belonging to any breakline.
- **Start**, **End** – When the start and end tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for an **End** tag. All intervening lines are assumed to belong sequentially to a breakline. If two **Start** tags are encountered without an intervening **End** tag, the break line is terminated and another started.
- **Start**, **Continue** – When the start and continue tags are used, each line of the data file is searched to indicate the initiation of a breakline. That is triggered when the **Start** tag is found. Every line thereafter is searched for a **Continue** tag. If any line is encountered without a **Continue** tag, the breakline is terminated and the vertex associated with that line is not included in a breakline.

Example of a tab delimited file using the following breakline tags:

- Start: 1
- Continue: 2
- End: 4
- Not in breakline: 5

```
xcoord        ycoord       zcoord   breakline_tag
215962.9      85203.098    1.483    1
215957.638    85193.069    1.483    2
215963.278    85184.35     1.483    2
215979.111    85179.328    1.483    2
216056.51     85209.371    1.483    4
215992.462    85201.477    7.034    1
216127.386    85264.681    7.034    2
216267.187    85327.936    7.034    2
216371.217    85381.431    7.034    4
219261.939    90247.944    8.763    5
219461.211    90220.556    9.167    5
219678.994    90179.064    9.468    5
```

Related Topics

- File Import Wizard
- Breaklines

Scatter Breakline Menu

The menu items operate on the active scatter set unless otherwise noted:

- **Force Breaklines –** force scatter triangle edges to follow the selected breaklines by swapping triangle edges. If no breaklines are selected, all breaklines will be forced.
- **Merge –** merge selected breaklines to form a single breakline. Available if more than one breakline is selected.
- **Split –** split a single breakline into multiple breaklines at the selected scatter vertex. Available if a scatter vertex is selected that is connected to a breakline.

Related Topics

- Scatter Module
- Scatter Module Menus
- Importing Scatter Breaklines

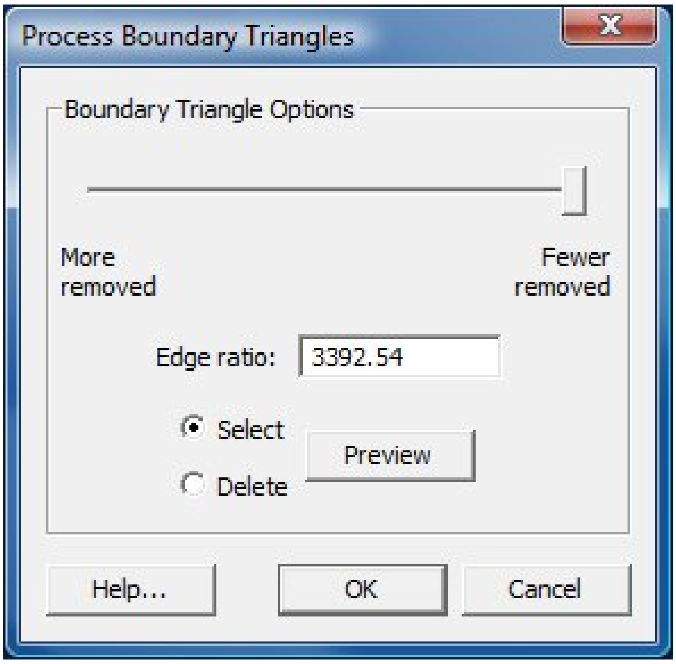
Process Boundary Triangles

When scatter points are triangulated, the resulting convex hull [1] often contains triangles outside the desired mesh boundary. The Process Boundary Triangles dialog was developed to help remove invalid boundary triangles. To open the Process Boundary Triangles dialog:

- In the Scatter Module, make the Select Triangle tool active
- Select "Process Boundary Triangles..." from the right mouse click menu

OR

- In the Scatter Module, select "Process Boundary Triangles..." from the Triangles menu

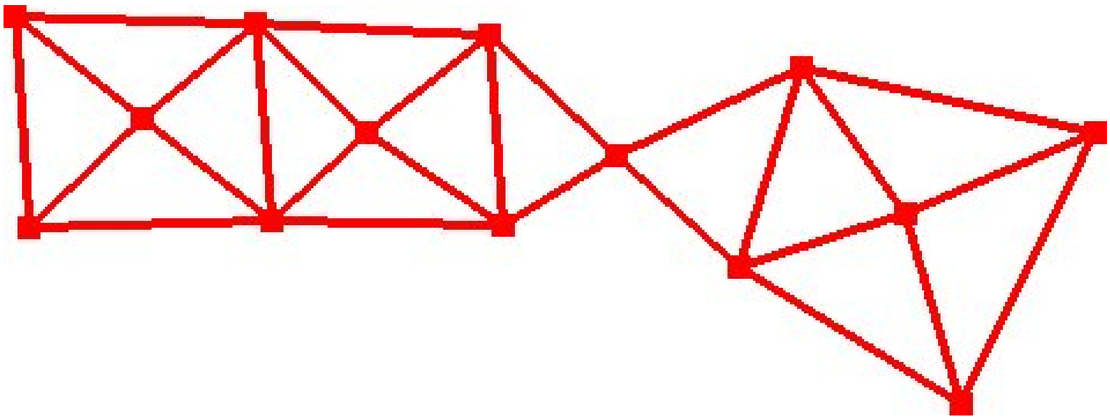


Removal Options

- Edge Ratio – A small edge ratio will result in the removal of more triangles than a large edge ratio. Specifying too small of an edge ratio will result in the removal of valid triangles. The edge ratio is calculated by dividing the length of the triangle edge on the boundary by the length of the smallest triangle edge connected to the scatter vertex end points of the triangle edge on the boundary.
- Select – Triangles meeting the specified edge ratio are selected when OK is pressed
- Delete – Triangles meeting the specified edge ratio are deleted when OK is pressed
- Preview – Preview which triangles will be processed based on the specified edge ratio

Using the Tool / Practical Notes

- Areas on the interior of the triangulation, such as islands, should be manually "seeded" by deleting one of the unneeded triangles in the interior region.
- Triangles will not be removed if their removal would result in:
  - the removal of a scatter vertex from the triangulation
  - the creation of a "bow tie" in the scatter set. A bow tie, a point in the scatter set where the scatter set is "pinched" to a single point, is shown in the image below.



Related Topics

- Select Thin Triangles

References

[1] http://en.wikipedia.org/wiki/Convex_hull

Interpolate to Scatter Set

This option interpolates one scattered dataset to another set. Two sets must exist for this option to be enabled. A dialog appears and the user selects the scatter set to interpolate from. That scatter set is interpolated to the active scatter set. The Options button brings up the Interpolation Options dialog, allowing the user to set the interpolation type. The interpolation uses an extrapolation value of 0.0.

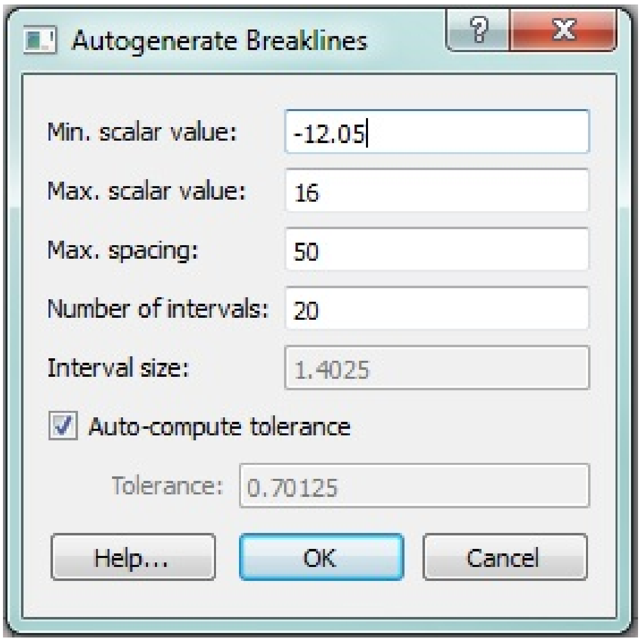
Related Topics

- Grid Frame Dialog
- Scatter Menu
- Scatter Data Menu
- Scatter Triangles Menu

Generate Contour Breaklines

Breaklines in scatter sets can greatly improve the representation of a physical surface. The Generate Contour Breaklines tool is used to create breaklines following specified dataset values. This option was designed to work on scatter data that has been digitized to follow features.

The following dialog is brought up by right-clicking on the scatter set and selecting "Generate Contour Breaklines..."



- Min. Scalar Value – The minimum of the range to be used for autogenerating breaklines (defaulted to the minimum scalar value of the scatter data)
- Max. Scalar Value – The maximum of the range to be used for autogenerating breaklines (defaulted to the maximum scalar value of the scatter data)
- Max. Spacing – The maximum distance allowed between two adjacent vertices in a breakline
- Number of Intervals – The number of dataset intervals from which the breaklines will be created
- Interval Size – The size of each interval (this is equal to the entire range - defined by the min. and max. scalar values - divided by the number of intervals)
- Auto-compute Tolerance – SMS will automatically compute the tolerance such that all datasets falling within an interval will be included in the breakline generation. The auto-computed tolerance is equal to half of the interval

size. This value can be overridden.

SMS creates the breaklines for each contour interval by gathering all vertices in the scatterset that are within the specified tolerance. Starting with the first vertex, SMS will search for the next closest vertex in the scatterset that is also within the specified tolerance. If the closest vertex is within the specified maximum spacing, it becomes the next vertex in the breakline. SMS continues adding vertices to the breakline until it cannot find a vertex within the spacing limit. At this point, SMS will end the breakline and begin to create another breakline using the same method.

3.10.c. Scatter Data Menu

Scatter Data Menu

Most of the SMS modules have a Data menu, but the items in this menu are different for each module. The menu items work with the active scatter set unless otherwise noted. The Scatter Module commands include:

Dataset Commands

- Data Calculator – Can be used to perform mathematical operations with datasets to create new datasets.
- Dataset Toolbox – Opens the *Data Toolbox* containing various tools to work with datasets.
- Create Datasets – Brings up the *Create Datasets* dialogue for creating functions for the active scatter set.
- Smooth Size Dataset – Opens the *Smooth Dataset* dialog that can be used to condition scattered data scalar values.
- Transform – Brings up the *Transform* dialog. The *Transform* command is used to move scatter points.
- Zonal Classification – Can be used to identify areas that meet a set of criteria. The criteria can be based upon scalar dataset values and/or specific material ids in a coverage. Opens the *Classification Wizard*.

Visualization Commands / Options

- **Contour Options** – Brings up the *Display Options* dialogue. See Contour Options for more information.
- **Vector Options** – Brings up the *Display Options* dialogue. See Vector Visualization for more information.
- **Film Loop** – Opens the *Film Loop Setup Wizard*. See Animation for more information.

Data Conversion Commands

- **Scatter → Mesh**

  The scatter points are converted to a mesh by this command. All functional data, scalar and vector, is copied to the mesh. The mesh nodes can be triangulated in the Mesh module using the menu command *Elements* | *Triangulate*.

- **Scatter Contour → Feature**

  When this command is invoked, the Create Contour Arcs dialog opens. For more information, see Create Contour Arcs.

- **Boundary → Feature**

  The outer boundary of the scatter set is converted to Map module arcs. Arcs are created where scatter triangles do not border another triangle.

Scatter Commands

- **Find**

  This menu item allows you to find a specific scatter point either by specifying its ID or by specifying the nearest (x,y) coordinate. This can be useful if one specific scatter point needs to be located in a large scatter set.

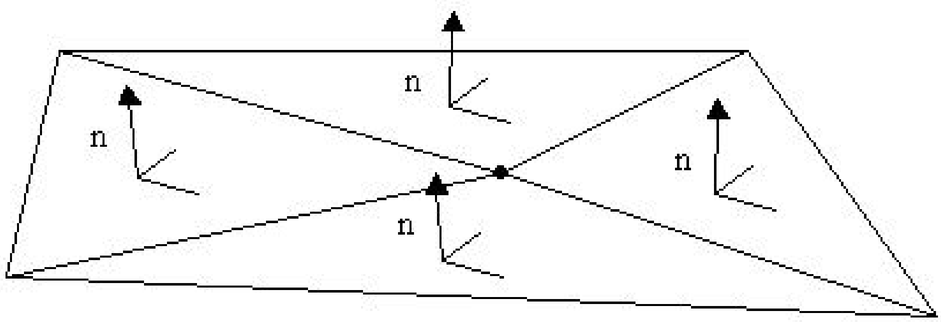
- **Scatter Filter**

  There are two filter options. The first option is to filter by adjacent triangle normal angle filter. The second option is to filter by using the VTK Decimate Pro.

  You can access the Filter Options dialog by going to *Data | Filter...* while in the Scatter Module.

  - **Filter – Adjacent Triangle Normal Angle Filter**

    Redundant and overlapping data may exist in a scatter. SMS offers the ability to filter the data and remove unnecessary data points in relatively flat areas in the Filter option from the Scatter module, Data menu. The user specifies an angle. Each data point is checked to see if it is in a flat region by dotting the normals of the surrounding triangles.



    If the normals are all within the specified angle, the region is flat and the point is deleted.

  - **Filter – VTK Decimate Pro**

    VTK Decimate Pro is a filter to reduce the number of triangles in a triangle mesh, forming a good approximation to the original geometry. To get a detailed description of the options to set when performing this kind of filter, please visit the website:

    http://www.vtk.org/doc/release/4.0/html/classvtkDecimatePro.html [1]

- **Select/Delete Duplicate Points**

  This menu item changes according to the option set in the Scatter Options dialog. In the options, the user can opt to select or delete duplicate points and set a tolerance. The check works by checking each point and selecting/deleting any point that is within the tolerance. Points with lower ids are checked first; the point with the higher id is therefore selected/deleted.

Related Topics

- Scatter Module
- Scatter Module Menus
- Create Coastline
- Coastline Files

References

[1] http://www.vtk.org/doc/release/4.0/html/classvtkDecimatePro.html

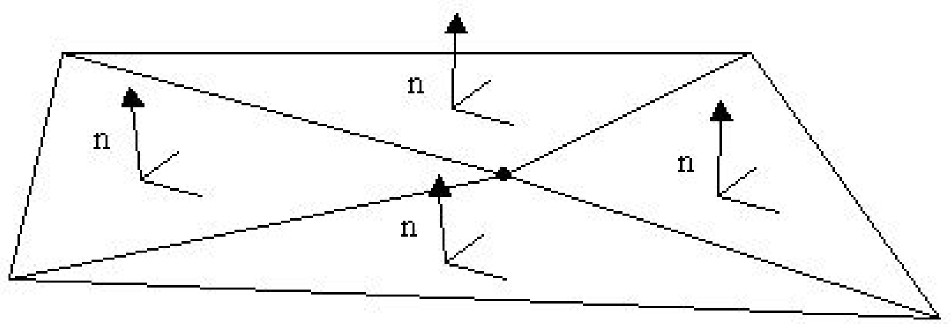
Scatter Filter

There are two filter options. The first option is to filter by adjacent triangle normal angle filter. The second option is to filter by using the VTK Decimate Pro.

You can access the Filter Options dialog by going to Data | Filter... while in the Scatter Module.

- **Filter – Adjacent Triangle Normal Angle Filter**

Redundant and overlapping data may exist in a scatter. SMS offers the ability to filter the data and remove unnecessary data points in relatively flat areas in the Filter option from the Scatter module, Data menu. The user specifies an angle. Each data point is checked to see if it is in a flat region by dotting the normals of the surrounding triangles.



If the normals are all within the specified angle, the region is flat and the point is deleted.

- **Filter – VTK Decimate Pro**

VTK Decimate Pro is a filter to reduce the number of triangles in a triangle mesh, forming a good approximation to the original geometry. To get a detailed description of the options to set when performing this kind of filter, please visit the website:

http://www.vtk.org/doc/release/4.0/html/classvtkDecimatePro.html [1]

3.10.d. Scatter Interpolation

Scatter Interpolation

Interpolation can be invoked explicitly or implicitly. The implicit invocation is part of the automatic mesh or grid generation. Explicit interpolation occurs when the user selects an **Interpolate to _____...** option in the Scatter menu in the Scatter Module. These commands require that at least one scatter set exist with at least one function associated. A mesh or grid must also exist in order for the associated interpolation command to be available.

Interpolation Dialog

When the user selects an interpolation command, the interpolation option dialog appears labeled to indicate what interpolation is being performed. The user selects the appropriate options and once the OK button is selected, the interpolation procedure is performed. The user specified options include:

- **New interpolated data set name** – The name of the new dataset (or function) created by interpolation.
- **Map Elevation** – For interpolation to mesh nodes, the new data set is mapped to be the elevation function.
- **Interpolation Method** – Since no interpolation scheme is superior in all cases, SMS supports three interpolation techniques. Many other methods are possible, however, since surface water modeling requires a fairly rich data set, the more simplistic interpolation methods are more applicable. The user selects a current method that is used for all interpolation until the user selects another method. The supported methods include:
  1. Linear
  2. Inverse Distance Weighted (IDW)
  3. Natural Neighbor (NN)
  4. Laplacian Interpolation (Cartesian grids only)
- **Default Extrapolation Value** – If the scatter set does not bound the data being interpolated to, the extrapolation value is used (for Linear and NN interpolation only).
- **Existing Dataset Value** – The corresponding value from a specified existing dataset can be used for locations outside of the bounds of the scatter set. The dataset must be from the same object being interpolated to and must be of the same type (i.e. scalar, vector).
- **Truncate Values** – When interpolating a set of values, it is sometimes useful to limit the interpolated values to a specific range. For example, when interpolating contaminant concentrations, a negative value of concentration is meaningless. However, many interpolation schemes will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by entering a minimum and maximum interpolation value.
- **Scatter Set/Function** – The user selects the desired scatter point set and the function to interpolate from.
- **Time Steps** – The user selects the option to interpolate a single time step or multiple time steps (if the function is transient).
- **Time Step Interpolation** – If interpolating multiple time steps, the user can select the beginning time, the step size, and the number of time steps to interpolate. The user can also choose to interpolate between time steps or to match time steps that fall within the specified time range.

Related Topics

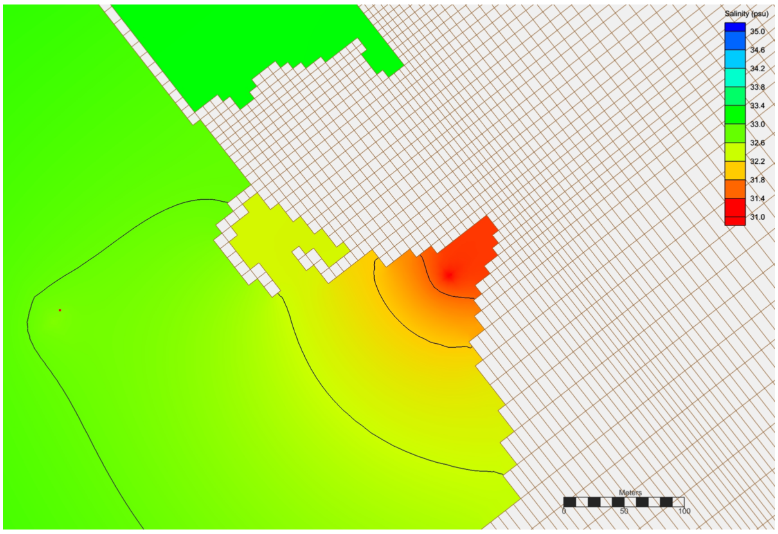
- Scatter Module
- Scatter Menu

Laplacian Interpolation

Laplacian interpolation is an interpolation/extrapolation approach that tries to give smooth gradients between areas of known values. You can only perform Laplacian interpolation on a cell-centered Cartesian grid.

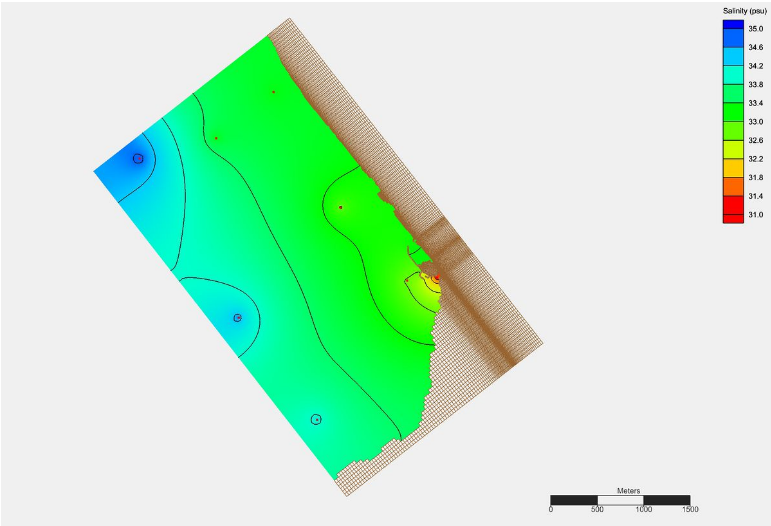
When you perform Laplacian interpolation, the values at each scatter vertex are considered known values and assigned to the Cartesian grid cells that they lie within. We will refer to the known locations/values as seed location/values. The seed values are held constant throughout the process. The algorithm assigns the average seed value to every cell in the grid. Once the initial values have been set, the algorithm repeatedly sweeps the grid smoothing the data values. The result gives the seed values at each of their locations and smooth transitions between the values. The algorithm will also extrapolate outside of the original seed values.

For models that distinguish between wet and dry cells (like CMS-Flow), the Laplacian interpolation will only be performed inside of wet (computational) cells. This gives the often desirable effect that the gradients will follow flow-paths. Cells that are nearby but do not have a water connection, will have a greater difference in value than cells that have a water connection. This effect can be seen in the image below.



Laplacian interpolation for salinity values

The picture below is of the same domain but zoomed out so you can see the full domain. Notice the smooth transitions between distinct values.



Laplacian interpolation for salinity full domain

Inverse Distance Weighted Interpolation

One of the most commonly used techniques for interpolation of scatter points is inverse distance weighted (IDW) interpolation. Inverse distance weighted methods are based on the assumption that the interpolating surface should be influenced most by the nearby points and less by the more distant points. The interpolating surface is a weighted average of the scatter points and the weight assigned to each scatter point diminishes as the distance from the interpolation point to the scatter point increases. Several options are available for inverse distance weighted interpolation. The options are selected using the Inverse Distance Weighted Interpolation Options dialog. This dialog is accessed through the Options button next to the Inverse distance weighted item in the 2D Interpolation Options dialog. SMS uses Shepard's Method for IDW:

Shepard's Method

The simplest form of inverse distance weighted interpolation is sometimes called "Shepard's method" (Shepard 1968). The equation used is as follows:

$$F(x,y) = \sum_{i=1}^{n} w_i f_i$$

where n is the number of scatter points in the set, fi are the prescribed function values at the scatter points (e.g. the data set values), and wi are the weight functions assigned to each scatter point. The classical form of the weight function is:

$$w_i = \frac{h_i^{-p}}{\sum_{j=1}^{n} h_j^{-p}}$$

where p is an arbitrary positive real number called the power parameter (typically, p=2) and hi is the distance from the scatter point to the interpolation point or

$$h_i = \sqrt{(x - x_i)^2 + (y - y_i)^2}$$

where (x,y) are the coordinates of the interpolation point and (xi,yi) are the coordinates of each scatter point. The weight function varies from a value of unity at the scatter point to a value approaching zero as the distance from the scatter point increases. The weight functions are normalized so that the weights sum to unity.

The effect of the weight function is that the surface interpolates each scatter point and is influenced most strongly between scatter points by the points closest to the point being interpolated.

Although the weight function shown above is the classical form of the weight function in inverse distance weighted interpolation, the following equation is used in SMS:

$$w_i = \frac{\left[\frac{R - h_i}{Rh_i}\right]^2}{\sum_{j=1}^{n}\left[\frac{R - h_j}{Rh_j}\right]^2}$$

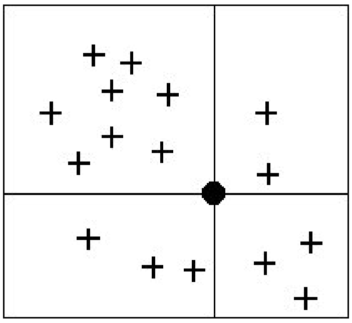
where hi is the distance from the interpolation point to scatter point i, R is the distance from the interpolation point to the most distant scatter point, and n is the total number of scatter points. This equation has been found to give superior results to the classical equation (Franke & Nielson, 1980).

The weight function is a function of Euclidean distance and is radially symmetric about each scatter point. As a result, the interpolating surface is somewhat symmetric about each point and tends toward the mean value of the scatter points between the scatter points. Shepard's method has been used extensively because of its simplicity.

Computation of Nodal Function Coefficients

In the IDW Interpolation Options dialog, an option is available for using a subset of the scatter points (as opposed to all of the available scatter points) in the computation of the nodal function coefficients and in the computation of the interpolation weights. Using a subset of the scatter points drops distant points from consideration since they are unlikely to have a large influence on the nodal function or on the interpolation weights. In addition, using a subset can speed up the computations since less points are involved.

If the Use subset of points option is chosen, the Subsets button can be used to bring up the Subset Definition dialog. Two options are available for defining which points are included in the subset. In one case, only the nearest N points are used. In the other case, only the nearest N points in each quadrant are used as shown below. This approach may give better results if the scatter points tend to be clustered.



The Four Quadrants Surrounding an Interpolation Point.

If a subset of the scatter point set is being used for interpolation, a scheme must be used to find the nearest N points. Two methods for finding a subset are provided in the Subset Definition dialog: the global method and the local method.

Global Method

With the global method, each of the scatter points in the set are searched for each interpolation point to determine which N points are nearest the interpolation point. This technique is fast for small scatter point sets but may be slow for large sets.

Local Method

With the local methods, the scatter points are triangulated to form a temporary TIN before the interpolation process begins. To compute the nearest N points, the triangle containing the interpolation point is found and the triangle topology is then used to sweep out from the interpolation point in a systematic fashion until the N nearest points are found. The local scheme is typically much faster than the global scheme for large scatter point sets.

Computation of Interpolation Weights

When computing the interpolation weights, three options are available for determining which points are included in the subset of points used to compute the weights and perform the interpolation: subset, all points, and enclosing triangle.

Subset of Points

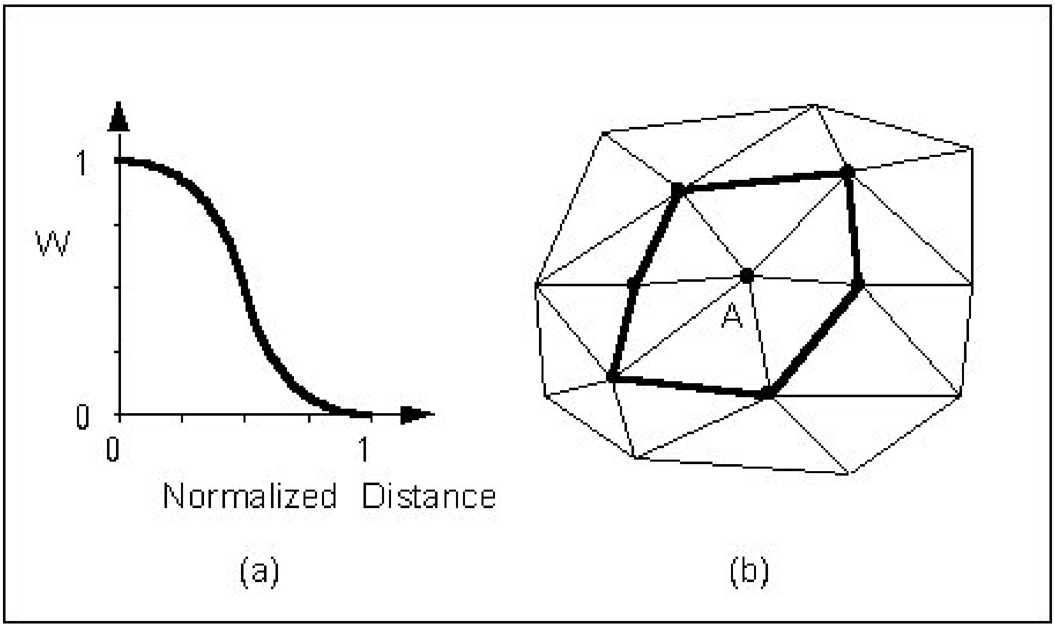
If the Use subset of points option is chosen, the Subset Definition dialog can be used to define a local subset of points.

All Points

If the Use all points option is chosen, a weight is computed for each point and all points are used in the interpolation.

Enclosing Triangle

The Use vertices of enclosing triangle method makes the interpolation process a local scheme by taking advantage of TIN topology (Franke & Nielson, 1980). With this technique, the subset of points used for interpolation consists of the three vertices of the triangle containing the interpolation point. The weight function or blending function assigned to each scatter point is a cubic S-shaped function as shown in part a of the figure below. The fact that the slope of the weight function tends to unity at its limits ensures that the slope of the interpolating surface is continuous across triangle boundaries.

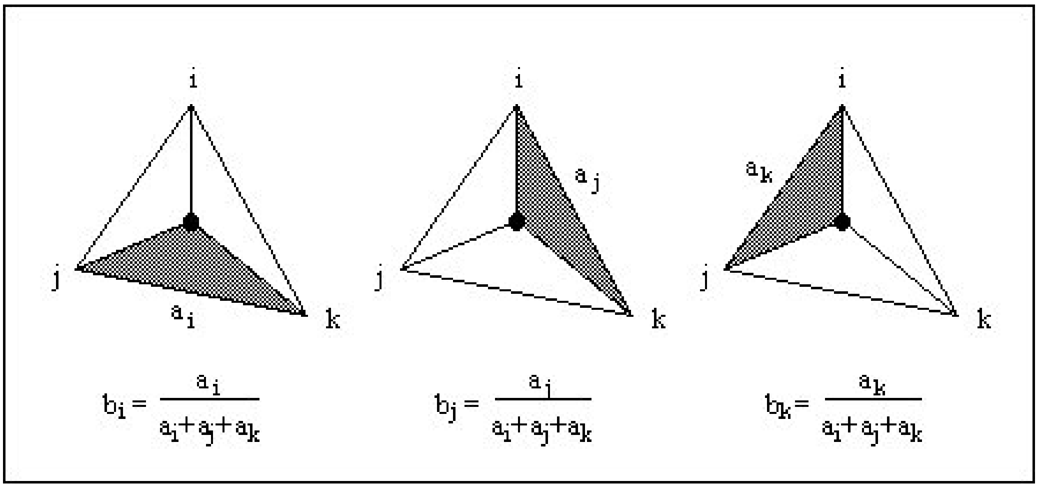


(a) S-Shaped Weight Function and (b) Delauney Point Group for Point A.

The influence of the weight function extends over the limits of the Delauney point group of the scatter point. The Delauney point group is the "natural neighbors" of the scatter point, and the perimeter of the group is made up of the outer edges of the triangles that are connected to the scatter point as shown in part b. The weight function varies from a weight of unity at the scatter point to zero at the perimeter of the group. For every interpolation point in the interior of a triangle there are three nonzero weight functions (the weight functions of the three vertices of the triangle). For a triangle T with vertices i, j, & k, the weights for each vertex are determined as follows:

$$w_i(x,y) = b_i^2(3-2b_i) + 3\frac{b_i^2 b_j b_k}{b_i b_j + b_i b_k + b_j b_k}\left\{ b_j\left[\frac{\|e_i\|^2 + \|e_k\|^2 - \|e_j\|^2}{\|e_k\|^2}\right] + b_k\left[\frac{\|e_i\|^2 + \|e_j\|^2 - \|e_k\|^2}{\|e_j\|^2}\right]\right\}$$

where $\|e_i\|$ is the length of the edge opposite vertex i, and b_i, b_j, b_k are the area coordinates of the point (x,y) with respect to triangle T. Area coordinates are coordinates that describe the position of a point within the interior of a triangle relative to the vertices of the triangle. The coordinates are based solely on the geometry of the triangle. Area coordinates are sometimes called "barycentric coordinates." The relative magnitude of the coordinates corresponds to area ratios as shown below:



Barycentric Coordinates for a Point in a Triangle.

The XY coordinates of the interior point can be written in terms of the XY coordinates of the vertices using the area coordinates as follows:

$$x = b_i x_i + b_j x_j + b_k x_k$$
$$y = b_i y_i + b_j y_j + b_k y_k$$
$$1.0 = b_i + b_j + b_k$$

Solving the above equations for bi, bj, and bk yields:

$$b_i = \frac{1}{2A}\left[\left(x_j y_k - x_k y_j\right) + \left(y_j - y_k\right)x + \left(x_k - x_j\right)y\right]$$
$$b_j = \frac{1}{2A}\left[\left(x_k y_i - x_i y_k\right) + \left(y_k - y_i\right)x + \left(x_i - x_k\right)y\right]$$
$$b_k = \frac{1}{2A}\left[\left(x_i y_j - x_j y_i\right) + \left(y_i - y_j\right)x + \left(x_j - x_i\right)y\right]$$
$$A = \frac{1}{2}\left(x_i y_j + x_j y_k + x_k y_i - y_i x_j - y_j x_k - y_k x_i\right)$$

Using the weight functions defined above, the interpolating surface at points inside a triangle is computed as:

$$F(x,y) = w_i(x,y)Q_i(x,y) + w_j(x,y)Q_j(x,y) + w_k(x,y)Q_k(x,y)$$

where w_i, w_j, and w_k are the weight functions and Q_i, Q_j, and Q_k are the nodal functions for the three vertices of the triangle.

Return to Scatter Interpolation

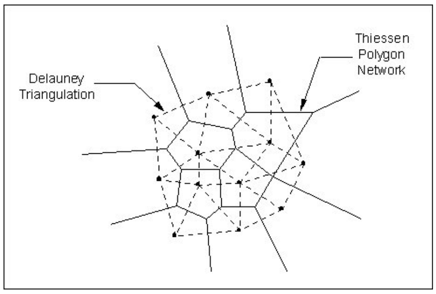
Natural Neighbor Interpolation

Natural neighbor interpolation is also supported in SMS. Natural neighbor interpolation has many positive features. It can be used for both interpolation and extrapolation and it generally works well with clustered scatter points. Natural neighbor interpolation was first introduced by Sibson (1981). A more detailed description of natural neighbor interpolation in multiple dimensions can be found in Owen (1992).

The basic equation used in natural neighbor interpolation is identical to the one used in IDW interpolation:

$$\lambda_m(n) = \frac{\kappa_m(n)}{\kappa(n)}$$

SMS uses nodal functions with IDW. The nodal function can be selected using the Natural Neighbor Interpolation Options dialog. The difference between IDW interpolation and natural neighbor interpolation is the method used to compute the weights and the method used to select the subset of scatter points used for interpolation.



Delauney Triangulation and Corresponding Thiessen Polygon Network for a Set of Scatter Points.

Natural neighbor interpolation is based on the Thiessen polygon network of the scatter point set. The Thiessen polygon network can be constructed from the Delauney triangulation of a scatter point set. A Delauney triangulation is a TIN that has been constructed so that the Delauney criterion has been satisfied.

There is one Thiessen polygon in the network for each scatter point. The polygon encloses the area that is closer to the enclosed scatter point than any other scatter point. The polygons in the interior of the scatter point set are closed polygons and the polygons on the convex hull of the set are open polygons.

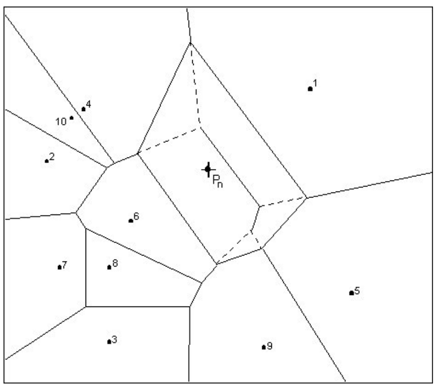
Each Thiessen polygon is constructed using the circumcircles of the triangles resulting from a Delauney triangulation of the scatter points. The vertices of the Thiessen polygons correspond to the centroids of the circumcircles of the triangles.

Local Coordinates

The weights used in natural neighbor interpolation are based on the concept of local coordinates. Local coordinates define the "neighborliness" or amount of influence any scatter point will have on the computed value at the interpolation point. This neighborliness is entirely dependent on the area of influence of the Thiessen polygons of the surrounding scatter points.

To define the local coordinates for the interpolation point, Pn, the area of all Thiessen polygons in the network must be known. Temporarily inserting Pn into the TIN causes the TIN and the corresponding Thiessen network to change, resulting in new Thiessen areas for the polygons in the neighborhood of Pn.

The concept of local coordinates is shown graphically in the following figure. Points 1–10 are scatter points and Pn is a point where some value associated with points 1–10 is to be interpolated. The dashed lines show the edges of the Thiessen network before Pn is temporarily inserted into the TIN and the solid lines show the edges of the Thiessen network after Pn is inserted.



Overlapping Thiessen Polygon Areas Used in Computation of Local Coordinates.

Only those scatter points whose Thiessen polygons have been altered by the temporary insertion of Pn are included in the subset of scatter points used to interpolate a value at Pn. In this case, only points 1, 4, 5, 6, & 9 are used. The local coordinate for each of these points with respect to Pn is defined as the area shared by the Thiessen polygon defined by point Pn and the Thiessen polygon defined by each point before point Pn is added. The greater the common area, the larger the resulting local coordinate, and the larger the influence or weight the scatter point has on the interpolated value at Pn.

If we define k(n) as the Thiessen polygon area of Pn and km(n) as the difference in the Thiessen polygon area of a neighboring scatter point, Pm, before and after Pn is inserted, then the local coordinate lm(n) is defined as:

$$\lambda_m(n) = \frac{\kappa_m(n)}{\kappa(n)}$$

The local coordinate lm(n) varies between zero and unity and is directly used as the weight, wm(n), in the interpolation equation. If Pn is at precisely the same location as Pm, then the Thiessen polygon areas for Pn and Pm are identical and lm(n) has a value of unity. In general, the greater the relative distance Pm is from Pn, the smaller its influence on the final interpolated value.

Related Topics

- Scatter Interpolation

Linear Interpolation

If the linear interpolation scheme is selected, the 2D scatter points are first triangulated to form a temporary TIN. If the surface is assumed to vary linearly across each triangle, the TIN describes a piecewise linear surface which interpolates the scatter points. The equation of the plane defined by the three vertices of a triangle is as follows:

$$Ax + By + Cz + D = 0$$

where A, B, and C, and D are computed from the coordinates of the three vertices (x1,y1,z1), (x2,y2,z2), & (x3,y3,z3):

$$A = y_1(z_2 - z_3) + y_2(z_3 - z_1) + y_3(z_1 - z_2)$$
$$B = z_1(x_2 - x_3) + z_2(x_3 - x_1) + z_3(x_1 - x_2)$$
$$C = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$$
$$D = -Ax_1 - By_1 - Cz_1$$

The plane equation can also be written as:

$$z = f(x,y) = -\frac{A}{C}x - \frac{B}{C}y - \frac{D}{C}$$

which is the form of the plane equation used to compute the elevation at any point on the triangle.

Since a TIN only covers the convex hull of a scatter point set, extrapolation beyond the convex hull is not possible with the linear interpolation scheme. Any points outside the convex hull of the scatter point set are assigned the default extrapolation value entered at the bottom of the Interpolation Options dialog.

Return to Scatter Interpolation

3.10.e. Data Sources

Historical Surveys

There are several sources of historic surveys. These include previous studies done by a modeler or company and compiled databases such as GEO-DAS or ETOPO. These data sources can be imported into SMS and used either as the basis of a finite computation domain (mesh or grid), or as a scattered dataset or DEM. Care must be taken into account the age and quality of the data and make sure all data sources are converted to a single coordinate system.

Return to Scatter Module

Electronic Charts

Since surveys can be expensive to obtain, and DEMs may not be applicable, another option available for the hydraulic modeler is the use of topographic/bathymetric charts or historic nautical charts. If these types of maps can be digitized into an electronic format, they can be read into SMS and displayed on the screen. The goal is to create a scattered data set from this electronic chart. The steps to do this include:

1. Scan the paper map and save it as an image (*.tif, *.jpg, ...).
2. Register the image (you may want to mark the map with your register points prior to scanning it).
3. Select the Create Vertex tool in the Data Module.
4. Digitize (click on the image on the screen) to create a vertex on a contour line in the image.
5. In the z edit box of the edit window set the z value to the contour value of the line.
6. Digitize along the specified contour value (the spacing of points along the contour lines should be approximately the same distance as the spacing between adjacent contours).
7. Repeat steps 4–6 for each contour line. Spot elevations can be entered by setting the z value to the value of the spot elevation and then creating a vertex at that location.
8. Triangulate the vertices once you are done.

This method becomes tedious for larger areas, but is ideally suited for smaller areas where there are not too many contours to be digitized.

Return to Scatter Module

3.11 Raster Module

Raster Module

At a glance

- Open and visualize raster data
- Supports many gridded elevation file formats. A complete list can be found at: www.globalmapper.com/product/formats.htm [1]
- One or more rasters are placed under a raster set in the project explorer.
- Convert raster to TIN (scatter set)
- Interpolate data from raster to TINs /2D Mesh/2D Grid
- Use rasters with observation profile plots

Rasters contain data (usually elevation) stored in pixels. Their resolution can vary, depending on the number of x and y cells the raster contains. The Raster Module allows us to open and visualize rasters of various formats and convert them into TIN (scatter sets), and interpolate their data into scatter sets, 2D meshes, and 2D grids.

Raster Sets

Each raster will be stored in a raster set. A raster set may contain multiple rasters. Ofter the data in one raster is also associated to another raster, when each raster is used to cover an area. When this is the case, you should group the raster's together under a single raster set. Raster sets are used in doing interpolation and plotting.

A new raster set can be created by right clicking on the root raster item in the project explorer, then select "New Raster Set". Existing raster's can be moved into a raster set by simply dragging the raster into the raster set.

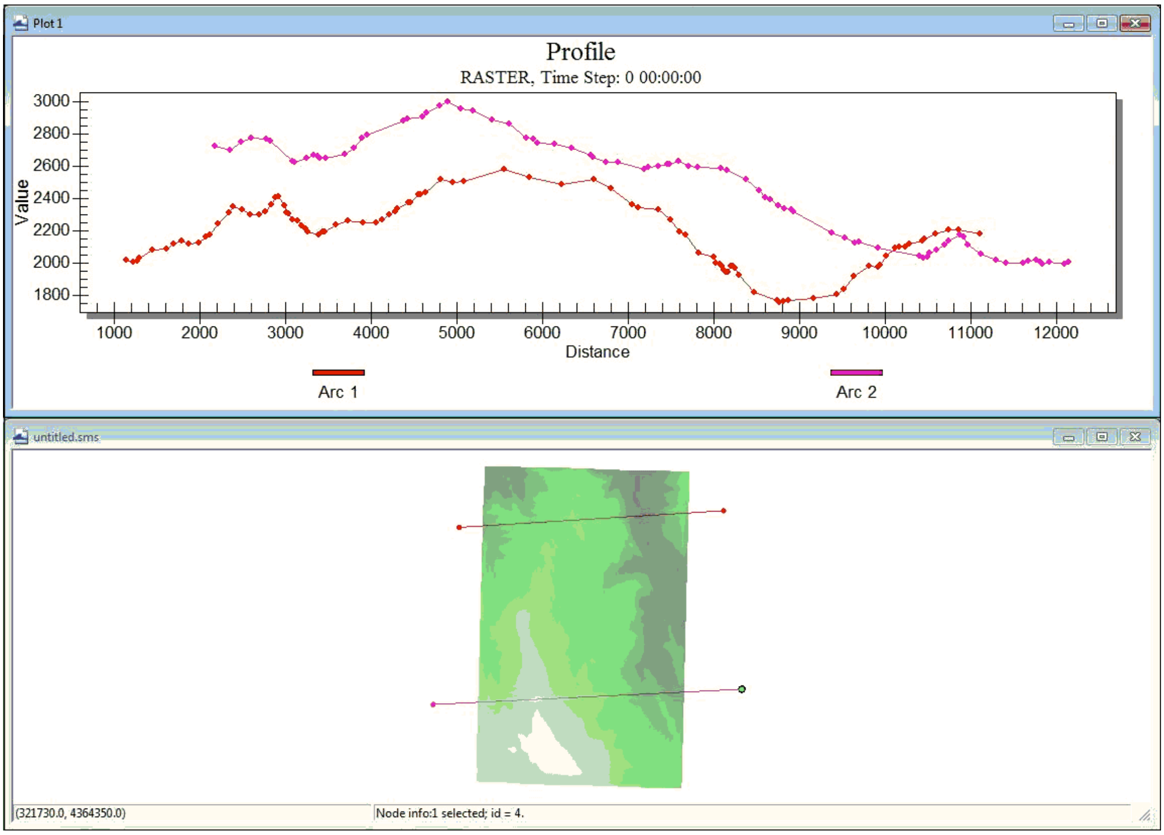
References

[1] http://www.globalmapper.com/product/formats.htm

Raster Functionalities

Profile Plot

SMS can generate many different types of 2D plots. Scalar data contained in visible rasters may be plotted using a specific type of 2D plot called a profile plot. When creating a profile plot for a raster using the Plot Wizard and 'specified data' is selected in Step 2, be sure to set the appropriate "Raster Set". Data will be plotted from the rasters in the selected raster set.



Interpolation

Values contained in rasters can be interpolated to existing scattersets, 2D meshes, and 2D grids. To interpolate the values from a raster, right click on the raster set in the project explorer and choose 'Interpolate'. Then choose the appropriate option for which object you wish to interpolate to.

Convert to Scatter

Values from rasters can also be converted into scatter data. This is done by right clicking on the raster item in the *Project Explorer* and selecting *Convert | Raster→2D Scatter*.

In special cases, one would want to convert only a selected portion of the raster to scatter data. For more information, see Raster Tools.

Raster Values as Elevation / Z Data

By default data associated with the raster is stored as elevation/z data. If a rasters projection is changed and the units change from meters to feet, the z values also would change. This becomes problematic if the raster values is used to represent NCLD land data and not elevation data. To keep NCLD land data from being projected, you can specify this in the raster set. This is done by right clicking on the raster set and selecting "Options..". A dialog appears allowing you to specify if the values are elevations or not elevations. If raster values are set as not elevations, when doing a projection the z values will remain unchanged.

Related Topics

- Plot Window

3.11.a Interface

Raster Module Interface

Clicking the raster module icon in the module tool bar will bring the raster module interface to display on the screen. The raster module interface can also be brought to display by clicking on a raster item in the project explorer. The raster module interface consists of the menu, tools, and project explorer right-click menus.

- Raster Menu Items
- Raster Tools
- Raster Right-Click Menus

Related Topics

- Project Explorer
- Toolbars

Raster Right Click Menus

Right Click Menu of "Raster Set" Project Explorer Item

Right clicking on the raster item in the Project Explorer will bring up the following options:

- Delete – deletes the current coverage.
- Rename – lets user rename the current coverage.
- Options – Clicking this will bring up a raster options dialog. This allows you to specify if your z values are elevations. If NCLD land data is being used, you should specify that raster values are not elevations. Then if you were to change projections, the z values that store your land data would not be affected.
- Interpolate – Data from raster sets may be interpolated to already existing TINs, meshes and 2D grids.

Right Click Menu of "Raster" Project Explorer Item

Right clicking on the raster item in the Project Explorer will bring up the following options:

- Delete
- Rename
- Convert – SMS allows rasters to be converted into TINs.
- Zoom to Raster

Right Click Menu of Dynamic Tools

See Raster Tools

Related Topics

- Project Explorer
- Dynamic Tools

Raster Menu Items

The menu bar that appears in the Raster Module:

- File – Has standard features as in other models.
- Edit – Has standard features as in other models.
- Display – Under the 'Raster' tab in 'Display Options', you can choose to 'Display as raster' or 'Display as surface'. 'Display as raster' is a 2D representation, while 'Display as surface' uses data from pixels as elevation to show a 3D surface. It should be noted that these display options affect performance. 'Display as raster' takes less memory and performs faster because it is a more efficient data structure. If 'Display as surface' is selected, you will be able to turn on or off contours, edges, or boundaries.
- Web – Has standard features as in other models.
- Window – Has standard features as in other models.
- Help – Has standard features as in other models.

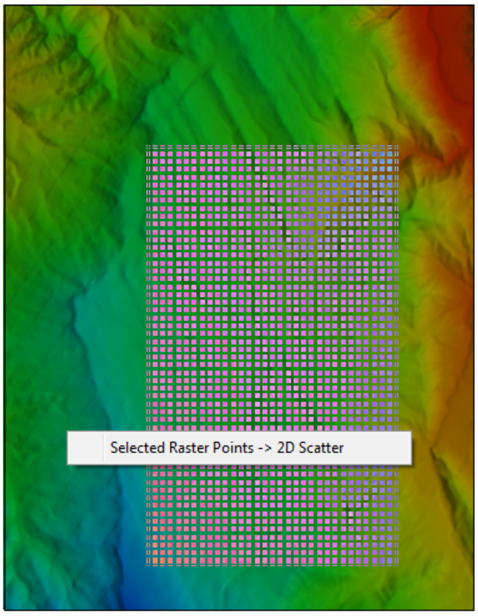
Related Topics

- Menu Bar

Raster Tools

The Raster Module contains only one dynamic tool called the 'Select Points' tool. Using this tool, one can edit/select one or more corners (points) of the pixels which make up the raster. Selecting cells with the 'Select Points' tool will show information about the points in the status bar.

Right Click Option



When one or more points are selected, one can right click in the graphics window, and the option to convert selected raster points to a scatter set appears. If this option is clicked, a new scatter set will appear in the project explorer.

Related Topics

- Help or Status Window
- Dynamic Tools

4. General Numeric Models

SMS Models

SMS provides pre- and post- processing for several numeric models. These models are developed and maintained by government or commercial entities rather than the developers of SMS.

Hydraulic Models

- ADCIRC (ADvanced CIRCulation Model) – Widely applied coastal circulation and coastal flooding model. Developed commercially.
- Coastal Modeling System CMS-FLOW – Suite of models that simulates a wide variety of coastal processes. Developed and maintained by the USACE.
- FESWMS – Developed in cooperation with the Federal Highway Administration (FHWA).
- Accessed through the generic model interface in cooperation with Aquaveo
  - SRH-2D – Developed at the United States Bureau of Reclamation.
  - RIVERFLO-2D – Commercially developed finite element model to route floods. Provides high resolution flood hydraulics.
  - HYDRO AS-2D – Developed commercially in Germany.
  - TUFLOW-FV – Finite volume model developed by the makers of TUFLOW (WBM).
- TABS – Suite of models for solving hydrodynamics and transport problems maintained by the USACE.
- TUFLOW – Finite difference model featuring combined 1D/2D models, very stable wetting drying, and advanced simulation management for Coastal, Riverine or Urban applications.

Wave Models

Harbor Design

- BOUSS-2D – Wave climate, circulation from waves, and sea state using the Boussinesque equations
- CGWAVE – Phase resolving wave reflection and refraction analysis

Wave Generation and Transformation

- STWAVE – Finite difference spectral wave energy model
- CMS-Wave (WABED)

Other Models

- Generic – Rather than a specific model interface, the "Generic" model interface in the mesh module is a collection of interface objects that can be configured by the user to generate specific types of data. The developers at Aquaveo work with some engine developers to utilize this tool. These engines are included in the list above. Other model developers are encouraged to contact Aquaveo for assistance in using these tools. The tools of the generic model interface can be utilized without coordination with Aquaveo. Se are aware of partial interfaces with the following:
  - FVCOM
  - HYDRO AS-2D

  - SRH-W
- PTM – Lagrangian Particle Tracking Model which tracks sediment particles based upon input hydrodynamics and wave effects.

Hydraulic Models (Feature Comparison)

Model	Riverine	Tidal Forcing	Wave Forcing	1D	2D	Hydraulic Structures	Sediment Transport	Advection / Dispersion
ADCIRC	no	yes	yes	no	yes	yes	no	no
CMS-Flow	no	yes	yes	no	yes	no	yes	no
FESWMS	yes	no	no	no	yes	yes	yes	no
TABS	yes	yes	no	yes	yes	yes	no	yes
TUFLOW	yes	yes	no	yes	yes	yes	no	no

Model Linkages / Steering

- General Steering
- RMA2 Spindown
- FESWMS Spindown
- CMS-Flow / CMS-Wave Steering
- ADCIRC / CMS-Wave Steering

Related Topics

- SMS Main page

FVCOM

FVCOM	
Model Info	
Model type	Prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model
Developer	Changsheng Chen University of Massachusetts-Dartmouth
Web site	FVCOM web site [1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

As stated on the FVCOM website (http://fvcom.smast.umassd.edu/FVCOM/index.html):

The Unstructured Grid **F**inite **V**olume **C**oastal **O**cean **M**odel (FVCOM) is a prognostic, unstructured-grid, finite-volume, free-surface, 3-D primitive equation coastal ocean circulation model developed by UMASSD-WHOI joint efforts. The model consists of momentum, continuity, temperature, salinity and density equations and is closed physically and mathematically using turbulence closure submodels. The horizontal grid is comprised of unstructured triangular cells and the irregular bottom is preseented using generalized terrain-following coordinates. The General Ocean Turbulent Model (GOTM) developed by Burchard's research group in Germany (Burchard, 2002) has been added to FVCOM to provide optional vertical turbulent closure schemes. FVCOM is solved numerically by a second-order accurate discrete flux calculation in the integral form of the governing equations over an unstructured triangular grid. This approach combines the best features of finite-element methods (grid flexibility) and finite-difference methods (numerical efficiency and code simplicity) and provides a much better numerical representation of both local and global momentum, mass, salt, heat, and tracer conservation. The ability of FVCOM to accurately solve scalar conservation equations in addition to the topological flexibility provided by unstructured meshes and the simplicity of the coding structure has make FVCOM ideally suited for many coastal and interdisciplinary scientific applications.

FVCOM was originally developed for the estuarine flooding/drying process in estuaries and the tidal-, buoyancy- and wind-driven circulation in the coastal region featured with complex irregular geometry and steep bottom topography. This model has been upgraded to the spherical coordinate system for basin and global applications. A non-hydrostatic version of FVCOM has been coded and is being tested.

Features

The present version of FVCOM includes:

- Choice of Cartesian or spherical coordinate system
- A mass-conservative wet/dry point treatment for the flooding/drying process simulation
- The General Ocean Turbulent Model (GOTM) modules (Burchard et al., 1999; Burchard, 2002) for optional vertical turbulent mixing schemes
- A water quality module to simulate dissolved oxygen and other environmental indicators
- 4-D nudging and Reduced/Ensemble Kalman Filters (implemented in collaboration with P. Rizzoli at MIT) for data assimilation
- Fully-nonlinear ice models (implemented by F. Dupont)

- A 3-D sediment transport module (based on the U.S.G.S. national sediment transport model) for estuarine and near-shore applications
- A flexible biological module (FBM) for food web dynamics study. With various pre-built functions and parameters for these groups, FBM allows users to either select a pre-built biological model (such as NPZ, NPZD, etc.) or to build their own biological model using the pre-defined pool of biological variables and parameterization functions. FBM includes seven groups:
  - Nutrients
  - Autotrophy
  - Heterotrophy
  - Detritus
  - Dissolved organic matter
  - Bacteria
  - Other.

FVCOM was originally coded for sigma-coordinates in the vertical and now has been upgraded to a generalized terrian-following coordinate system with choices of various topographic-following coordinates. FVCOM is written with Fortran 90 with MPI parallelization, and runs efficiently on single and multi-processor machines.

FVCOM is an open source code ocean community model that always welcomes new users. **This program is only permitted for use in non-commercial academic research and education.** Users are required to register [2] to receive the source codes, demo examples, and user manuals as well as some recommended postprocessing tools.

Graphical Interface

FVCOM uses the Generic Model Graphical Interface.

External Links

- FVCOM Model Developer Website [1]
- FVCOM Publications [3]
- FVCOM Forum [4]
- FVCOM Help [5]

Related Topics

- Generic Model Interface

References

[1] http://fvcom.smast.umassd.edu/FVCOM/index.html
[2] http://fvcom.smast.umassd.edu/FVCOM/Source/code.htm
[3] http://fvcom.smast.umassd.edu/FVCOM/FVCOMPubs/index.html
[4] http://www.dgeo.udec.cl/FVCOM/index.php
[5] http://fvcom.smast.umassd.edu/FVCOM/FVCOMHelp/fvcom_help.html

4.1. Generic Mesh Model

Generic Mesh Model

While SMS contains several custom interfaces for specific numerical models, it is not possible to build interfaces in SMS for every model in existance. SMS has a generic interface whereby any two-dimensional finite element or finite difference model can be run using SMS as a pre- and post-processor.

Functionality

The following can be defined for a generic model:

- Global Model Parameters
- Boundary Conditions
- Material Properties

With the model attributes defined, the user can generate a mesh or grid using the geometric tools in the Map, and Mesh or Cartesian Grid Modules. Model parameters can then be set, boundary conditions specified, and material properties applied to the geometric data and saved out to a generic file format.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the Generic Model. See Generic Model Graphical Interface for more information.

The Generic Model Graphical Interface contains tools to create and edit a Generic Model simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to Generic. If a mesh has already been created for a Generic Model simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to Generic. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the Generic Model Menu. See Generic Model Graphical Interface for more information.

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use the Generic Model Interface in SMS:

- General Section
  - Mesh Editing
  - Observation
- Models Section
  - Generic Mesh Model

Related Topics

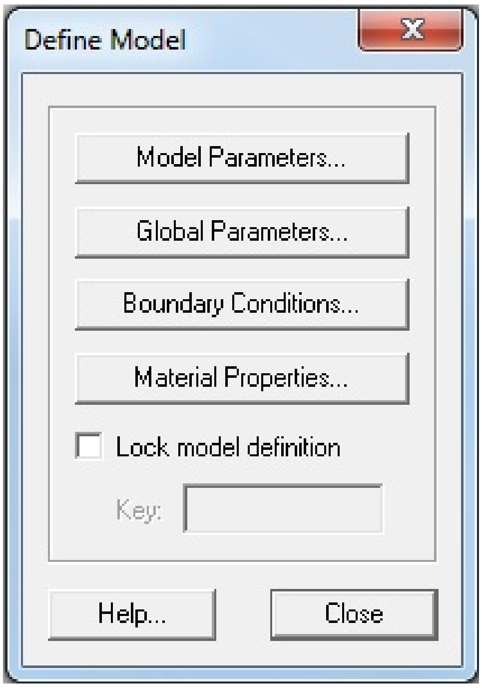
- BASEMENT
- HYDRO AS-2D
- SRH-2D
- ELCIRC
- FVCOM

Define Model Dialog

The Define Model Dialog is used to customize and define the model interface parameters that define various states and characteristics of a model. These model parameters may include items such as those needed to describe flow, channel roughness, and control structures. The parameters, names, and ranges can be created and customized by the user.

Parameters are organized into groups and are given suitable value ranges depending on the purposes of the model. Proper organization of parameters will increase the abilities of SMS as an interface.

Using the Define Model dialog the model interface can be renamed by the user. The Define Model dialog can be accessed when the Mesh Module is the active module.



Define Model Dialog for the Generic Model

Related Topics

- Mesh Module
- Generic Model Graphical Interface

Generic 2D Mesh Arc Attributes Dialog

The Generic 2D Mesh Feature Arc Attributes dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

- Arc Type
  - None
  - Boundary Conditions – Options button opens the Mesh Nodestring Boundary Conditions dialog

Related Topics

- Feature Objects Menu

Generic 2D Nodal BC, Nodestring and Element Display Options

The properties of all Gen2DM (Generic 2D Mesh) node boundary conditions, nodestrings, and elements that SMS displays on the screen can be controlled here. This window is accessible only if the current 2D Mesh model is Generic (Mesh module’s Data | Switch Current Model… menu command). Open Display Options, turn on either Nodal BC, Nodestrings, or Elements and then select the corresponding Options button.

Display options are available for all boundary condition entities defined in the Gen2DM model definition. If no BC entities have been defined then a message will state that fact and display options will not be shown. Each defined BC entity has a checkbox to toggle the display of the item and an Attribute Options button to adjust style and color.

Below the list of BC entities is the display options for Inactive entities. Inactive entities are BC entities that are associated with an inactive parameter group (activation is controlled via the Gen2DM | Global Parameters menu item).

The BC entities can be displayed with labels beside them. Labels allows the user to adjust the label font, style, size, color and whether they are displayed. When Labels is checked for display, the auxiliary Show BC Values in labels is available. If checked, BC entities’ values will be displayed following the entities’ description.

All on checks all BC entity display options.

All off unchecks all BC entity display options.

Related Topics

- Generic Model Graphical Interface

Generic Model Files

SMS stores all of the Generic Model input parameters in a 2D Mesh File (*.2dm). This 2D Mesh file includes:

- The mesh
  - Node locations
  - Element type
    - Triangle or quadrilateral
    - Linear or quadratic
  - Element connectivity
- Global parameters
- Boundary conditions
  - Nodal boundary conditions
  - Nodestring boundary conditions
  - Element boundary conditions
- Material properties

For more information, see the 2D Mesh File (*.2dm) article.

Related Topics

- HYDRO AS-2D
- SRH-W

Generic Model Graphical Interface

The Generic Model Graphical Interface includes tools to assist with creating, editing and debugging a Generic Model. The Generic Model interface exists in the Mesh Module.

Define Model

The Define Model Dialog is used to setup the options that apply to the simulation as a whole.

- Model Parameters
- Global Parameters
- Boundary Conditions
- Material Properties
- Lock Model Definition with Key

Boundary Conditions

All numeric models require boundary condition data. Generic Mesh Model boundary conditions can be defined on nodestrings, nodes, and elements. An entity (nodestring, node, or element) may have multiple boundary conditions set. To add or remove a boundary condition, select a node, nodestring, or element. Then right click and select assign BC. A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are active boundary conditions. Toggle the tree items as desired to activate/deactivate boundary conditions. The settings in Display→Display Options will determine how the boundary conditions are displayed.

The model developer can define boundary conditions as constant or as dynamic. This is stored in the BD card as the next to last field. The user can also assing multiple boundary conditions to nodes, nodestrings, and elements.To add or remove a boundary condition, select a node, nodestring, or element. Then right click and select assign BC. A dialog with a tree item appears listing all possible boundary conditions. Checked tree items are assigned. Toggle the tree items as desired to assign/unassign boundary conditions. The settings in Display→Display Options will determine how the boundary conditions are displayed.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Material Parameter Groups are the same as the Global Parameter Groups. To add a new Material Parameter Group, add it in the Global Parameter Group by "Gen2DM" → "Define Model" → "Global Parameters".

Designer /User can choose whether to have a single material group or multiple (1 for each group) by "Gen2DM" → "Define Model" → "Material Properties" → select /unselect "Have a seperate material assignment for each parameter group".

- Multiple
  - User can change the active group by "Gen2DM" → "Set Active Material Group".
  - Changing the "active material group" effects the display and assignment of materials.
  - In the "Gen2DM" → "Material Properties" dialog, only the active/assigned materials are displayed.
- Single
  - "Set Active Material Group" menu is hidden under the "Gen2DM" menu
  - In the "Gen2DM" → "Material Properties" dialog, all materials are displayed. The tabs across the top correspond to the Groups.

Dependencies

The generic model interface lets a generic model designer create a custom interface by setting up input parameters for their model.

The parameters may be global parameters, bc parameters (applied to node, element, or nodestring), or material parameters.

It is often useful to have certain parameters displayed only in some situations.

For example, the numeric engine may support manning values by depth or a single manning value for all depths. If the user chooses to use manning values by depth he or she would need to provide a curve for manning values based upon depth. If the user chooses to do a single manning value he or she would need to provide a single manning value.

This manning example could be expanded another level. Suppose the manning value mentioned above was part of the material properties for each material. Also, suppose that the engine supported chezy as well as manning to represent roughness but this had to be applied at a global level. If the global parameter is chezy, the material properties dialog should only show the option to enter a chezy value. If the global parameter is manning, the material properties dialog would let the user choose whether to provide this using a single value or a curve by depth. The controls would work as described in the preceding paragraph.

The generic model designer can use dependencies to accomplish both situations above. Dependencies show/hide parameters based upon the setting of a parent parameter. The parent parameter must be before the child parameter. If the child parameter is a material or boundary condition parameter, the parent can also be a global parameter.

Dependencies are controlled on the child level when defining the model. To assign a dependency the parent and child parameters must exist and the parent must have its options defined. Dependencies are specified by clicking on the "None" button in the dependency column for the child parameter. You specify the parent you want to use for this parameter whether a global parameter or local (whatever level you currently are defining). Then check the boxes next to the parent parameters that will allow the child parameter to be visible. A child option may be visible for several parent options.

Whenever a parent object is invisible all children that are dependent upon the option are also invisible.

Overview

The designer/user can define whether the curves in their model are interpolated to the timestep duration or not.

Saving the Model

The Generic Model Files are written automatically with the SMS project file or can be saved separately using the *File* | *Save Mesh* or *File* | *Save As* menu commands. See Generic Model Files for more information on the files used for the Generic Model run.

Generic Model Menu

See Generic Model Menu for more information.

Related Topics

- Generic 2D Nodal BC, Nodestring and Element Display Options
- Mesh Module

4.2. PTM

PTM

PTM	
Model Info	
Model type	Lagrangian particle tracker designed to allow the user to simulate particle transport processes.
Developer	Neil J. MacDonald, Ph.D. Michael H. Davies, M.Sc., Ph.D., P.Eng. Coldwater Consulting Ltd [1]
Web site	PTM web site [2]
Tutorials	Models Section • PTM (Pending)

The Particle Tracking Model (PTM) is a Lagrangian particle tracker designed to allow the user to simulate particle transport processes. PTM is funded through two US Army Corps of Engineers Engineering Research and Development Center (ERDC) research programs, the Coastal Inlets Research Program (CIRP) and the Dredging Operations and Environmental Research (DOER) Program.

Functionality

PTM has been developed for application to dredging and coastal projects including dredged material dispersion and fate, sediment pathway and fate, and constituent transport. The model contains algorithms that appropriately represent transport, settling, deposition, mixing, and resuspension processes in nearshore wave/current conditions. It uses waves and currents developed through other models and input directly to PTM as forcing functions.

Using the Model / Practical Notes

- The horizontal and vertical coordinates used for all PTM input files must be in meters. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.
- If last step trap is checked, the traps will not become active until the last time step.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the PTM model. See PTM Graphical Interface for more information.

The PTM Graphical Interface contains tools to create and edit an PTM simulation. The simulation consists of a geometric definition of the sources, traps, and a set of numerical parameters. The parameters define the hydrodynamic input and options pertinent to the model.

The interface is accessed by selecting the Particle Module and setting the current model to PTM. If a simulation has already been created or an existing simulation read, the particle object will exist in the Project Explorer and selecting that object will make the Particle module active and set the model to PTM. See the Particle Module documentation for guidance on visualizing results.

The interface consists of the Particle Module Menus and tools augmented by the PTM Menu. See PTM Graphical Interface for more information.

Theoretical Basis / Mathematical Details

Please refer to the model developer provided documentation listed in the external links section.

General Steps to Build a PTM Model

1. Open SMS
2. Although not required, it is generally easier to visualize your model if you read in the hydrodynamic solution
3. Convert the horizontal and vertical coordinates to meters. PTM cannot be run in Geographic Coordinates since it is a latitude/longitude system defined in decimal degrees.
4. Switch to the Map Module
5. Create a PTM type coverage
6. Create sources and traps using feature points, arcs, and polygons
7. Switch to the Particle Module
8. Use the PTM Menu to create a new PTM simulation
9. Use the PTM Menu to open the PTM model control dialog
    1. Specify input and output parameters in the model control
    2. Click on the "Create input file(s) from data" button to create a native sediment file, and hydrodynamic input files if needed
10. Save the simulation
11. Use the PTM Menu to perform a model check
12. Use the PTM Menu to run PTM

General Post Processing Steps

1. Use the display options to visualize the results
2. Use the Data Calculator or Create Data Sets Dialog to create particle data sets
3. Create a Cartesian Grid and use the Create Grid Data Sets Dialog to create Cartesian grid data sets
4. Review the Trap Output files in an ASCII file viewer

External Links

- U.S. Army Corps of Engineers DOER PTM website [2]
- Sep 2006 ERDC/CHL TR-06-20 PTM: Particle Tracking Model Report 1: Model Theory, Implementation, and Example Applications [3]
- Jul 2005 ERDC TN-DOER-D4 Particle Tracking Model (PTM) in the SMS: I. Graphical Interface [22]
- Jul 2005 ERDC TN-DOER-D5 Particle Tracking Model (PTM): II. Overview of Features and Capabilities [23]
- Jul 2005 ERDC TN-DOER-D6 Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples [24]
- Jul 2008 ERDC/CHL CHETN-IV-71 Particle Tracking Model (PTM) in the SMS 10: IV. Link to Coastal Modeling System [4]
- 2007 The Particle Tracking Model: Description and Processes [5]
- Application of the Particle Tracking Model to Predict the Farfield Fate of:
    - 2007 Sediment Suspended by Nearshore Dredging and Placement, Brunswick, GA [6]
    - 2008 Dredged Suspended Sediment at the Willamette River [7]
- 2009 Assessment of Dredging-Induced Sedimentation on Winter Flounder Spawning habitat [8]

Related Topics

- Particle Module
- Mesh Module

References

[1] http://www.coldwater-consulting.com
[2] http://el.erdc.usace.army.mil/dots/doer/ptm.html
[3] http://el.erdc.usace.army.mil/dots/doer/pdf/TR-06-20.pdf
[4] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-71.pdf
[5] http://el.erdc.usace.army.mil/dots/doer/pdf/LackeyandMcDonald_2007.pdf
[6] http://el.erdc.usace.army.mil/dots/doer/pdf/GAILANIetal_2008.pdf
[7] http://el.erdc.usace.army.mil/dots/doer/pdf/LackeyandSmith_2008.pdf
[8] http://el.erdc.usace.army.mil/dots/doer/pdf/LackeyKimClarke_WEDA.pdf

PTM Overview

PTM

Overview

The Particle Tracking Model (PTM) is a Lagrangian particle tracker designed to allow the user to simulate particle transport processes. PTM is funded through two US Army Corps of Engineers Engineering Research and Development Center (ERDC) research programs, the Coastal Inlets Research Program (CIRP) and the Dredging Operations and Environmental Research (DOER) Program. PTM has been developed for application to dredging and coastal projects including dredged material dispersion and fate, sediment pathway and fate, and constituent transport. The model contains algorithms that appropriately represent transport, settling, deposition, mixing, and resuspension processes in nearshore wave/current conditions. It uses waves and currents developed through other models and input directly to PTM as forcing functions.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the PTM model. See PTM Graphical Interface for more information.

The PTM Graphical Interface contains tools to create and edit an PTM simulation. The simulation consists of a geometric definition of the sources, traps, and a set of numerical parameters. The parameters define the hydrodynamic input and options pertinent to the model.

PTM Menu

The following menu commands are available in the PTM Menu:

Command	Functionality
New Simulation	Creates a new PTM simulation and adds it to the Project Explorer.
Model Check	Checks the active PTM simulation for common input errors.
Model Control	Opens the Model Control dialog (used to organize input files, specify model parameters, choose output options, etc.).
Run Model	Launches the PTM model using the active PTM simulation input files.

How do I use PTM?

The SMS tutorials are a good place to start learning to use SMS and associated models. The following tutorials will help get you started using TUFLOW.

Tools

PTM sources and traps are created in the Map Module as feature objects using the Map Module Tools.

Numeric engine background

Please refer to the model developer provided documentation listed in the external links section.

Related Topics

- PTM Arc Attributes Dialog
- PTM Control File
- PTM Feature Point Attributes Dialog
- PTM Gage Coverage
- PTM Graphical Interface
- PTM Menu
- PTM Model Check
- PTM Model Control
- PTM Particle Filters
- PTM Polygon Attributes Dialog
- PTM Sediment File
- PTM Source File
- PTM Sources
- PTM Trap File
- PTM Traps
- PTM Wave File
- Particle Module
- Mesh Module

PTM Model Control

The PTM model control is used to create the program control file, which contains general simulation options. Refer to the PTM manual[SMS:PTM#External_Links] for a more detailed description of how these parameters affect the model results.

Time

The time tab is used to specify time information to be written to the program control file. If a control on the dialog is not enabled, the corresponding keyword will not be written to the program control file. Controls on the dialog are disabled based on the options you have chosen on the other model control tabs. For example, if no trap file is specified on the files tab, the time controls related to traps will not be available. There is a plot on the dialog of the specified times.

Files

The files tab is used to specify input and output file information to be written to the program control file.

Pressing the "Create input file(s) from data..." button on the Files tab will open the Create PTM External Input Files dialog.

Native Sediment Grain Size

1. If "Uniform bed" is selected from the "Options" column an "Options" button is displayed under "Filename". Pressing this button allows the user to create uniform grain sizes for D35, D50 and D90. If the project is saved, the :SEDIMENT_FORMAT card is created, with the keyword UNIFORM and the 3 grain sizes being written out.
2. If Sediment file is selected, the user can manually select the filename by pressing the button under "Filename".
3. Pressing the "Create input file(s) from data" and selecting to create a Native sediment grain size file will automatically toggle the native sediments grain size to "sediments file" and place the new file under the "Filename".

Computations

The computations tab is used to specify model computation options to be written to the program control file.

Output

The output tab is used to specify model output options to be written to the program control file.

PTM Particle Filters

Particle Filters

The particle filters allow the user to evaluate specific particles based upon different criteria. The filters act exclusively upon particle datasets. Multiple filters can be used together to filter multiple datasets.

The filters affect:

- Displayed particles
- Selectable particles (hence selected particle info)
- Compute Grid datasets functionality

Dialog Description

The following controls are used to define the particle filters.

Filters

Use the New and Delete buttons to create / remove filters. Each filter may be enable / disabled by clicking its toggle box. The filters are defined by selecting a Dataset whose values will be compared against the operands / operators pairs defined in the Options control.

Datasets

Each filter operates on a single Particle Module dataset. Examples of datasets in the Particle Module include solutions created by the PTM model or using the data calculator.

Options

- Operators – Comparison operators

<	Less than
<=	Less than or equal to
equal	Equal to
not equal	Not equal to
>	Greater than
>=	Greater than or equal to

- Operands – Values compared against the selected dataset using the corresponding operators
- Conjunction – This toggle box disables the second operator / operand pair or performs the and function on them against the first operator / operand pair.

PTM Polygon Attributes Dialog

Feature polygons created in a PTM type coverage can be used to create either sources or traps.

Related Topics

- PTM Sources
- PTM Traps

PTM Traps

Overview

A trap is defined as an area into which particles enter and are counted.

Global Trap Options

Global options affect all traps.

- Time limits can be specified on the trap's operation in the PTM Model Control, Time Tab.
- Traps can be set to only be active on the last time step of the PTM simulation by selecting "Last timestep trap" in the PTM Model Control, Time Tab.

Individual Trap Options

Individual options are specified on a trap by trap basis.

- The trap may be open (particles are free to leave) or closed.
- Particles can be counted once per simulation (single trap) or every time they re-enter an open trap.
- The trap can have a bottom and top elevation defining the portion of the water column the trap exists within.
- ID – Integer value to identify the trap
- Name – Text to identify the trap

Horizontal Line Traps

Horizontal line traps lie in a vertical plane, defined by the trap end points and a top and bottom elevation.

Horizontal line traps have a direction associated with them. The direction of a trap determines which particles the trap will count.

- Decreasing x-coordinate – only counts particles passing through the trap moving in the negative x-direction.
- Either direction – counts particles passing through the trap regardless of direction.
- Increasing x-coordinate – only counts particles passing through the trap moving in the positive x-direction.

Horizontal Polygon Traps

Horizontal polygon traps lie on a horizontal plane and are extruded in the vertical direction based on the trap's top and bottom elevation.

Related Topics

- PTM Trap File
- PTM Sources

4.2.a. PTM Coverages

PTM Coverages

In the SMS interface, two types of coverages can be applied to a PTM simulation. These include:

- The PTM Source/Trap Coverage: used to define input for a PTM simulation.
- The PTM Gage Coverage: used to view time series of output data computed by the PTM engine.

PTM Gage Coverage

A "PTM Gage" coverage has been added to allow the creation of virtual gages. A virtual gage can be associated with either a point or a polygon. When a virtual gage is associated with a point, the user also specifies a radius that defines the area being monitored by the gage (creating a circle).

The gages can be thought of as open traps which monitor the number parcels and material that enter the specified region during the simulation.

When the user invokes the Particle Module Compute Grid Datasets command, the user can choose to compute datasets associated with a grid, or with a PTM Gage coverage. When the gage coverage option is selected, SMS computes output data associated with each point and/or polygon. This data consists of sets of xy series curves organized by solution and dataset name. Each curve represents a quantity related to the specific gage. These quantities include count, accumulation, concentrations, rate of exposure and all of the options supported by the compute dataset command including the options to bin concentration, exposure and dosage in the "Z" direction (creating 3D data). The output data is visualized using a PTM Gage plot type in the plot wizard for 2D Plots.

The polygons and points in the virtual gage coverage are created with discrete color attributes. Each point has a radius attribute that represents the area over which the computations will apply.

PTM Trap Output as Gages

In addition to creating virtual point and polygon gages, the PTM Gage Coverage can also be used to view data output by PTM in its Trap Output file. This functionality does not include the concentration, dosage and exposure options, but it does allow the user to visualize data related to closed traps.

4.2.a.1. PTM Sources

PTM Sources

Overview

The material which is to be modeled in PTM is released from sources. The amount of material released from each source is specified as a mass, either as an instantaneously released total mass, or as a mass release rate over a given time period. PTM represents this mass by a finite number of particles.

Particles can be introduced via three different types of sources:

- Point Sources
- Line Sources
- Area Sources

There can be any number of any source type used in a simulation, and different source types can be specified in the same simulation.

Point Sources

Instantaneous

If the material to be modeled is to be released at a single point in time, then an instantaneous source should be specified. An example of this type of release is an accidental spill from a vessel. This type of release occurs at a fixed location, and the full release of material occurs at the time given and with the properties specified.

Varying-Release

If the release of material occurs over a period of time, then a varying-release point source should be specified. An example of this type of release is a leak from a pipeline. The characteristics of release point sources can vary with time (e.g., release rate, three dimensional positions, etc.). Varying-release point sources can be started, stopped, re-started, moved, etc., as directed in a source release schedule, which is developed through the SMS interface.

The horizontal and vertical radii of both types of point sources can be specified in the source release schedule. If either radius is greater than zero, then the initial locations of the individual particles are varied so as to produce a two- or three-dimensional Gaussian-distributed cloud.

Line Sources

Line sources must either be vertical or horizontal and are varying-release. Particles released from a line source will have a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line. Line sources are specified by their end points. Line sources may move or change length, position, or discharge properties with time. Linear interpolation in time is used for most properties in a line source, but the characteristics of the release do not vary along the line source (e.g., the release rate of particles can vary with time for a line, but the rate will be the same over the length of that line). To model a line source with varying characteristics along the line, one could use a series of lines positioned end to end, each with different characteristics.

Vertical Line Source Datums

The following vertical line source datums can be used to specify the vertical line source top and bottom elevation:

- Bed (default) – Elevations are specified relative to the bed. The rate is given in kg/m/s by the user.
- Surface – Elevations are specified relative to the water surface. The rate is given in kg/m/s by the user.
- Depth distributed – Elevations are specified as percentages of the water depth. The rate is given in kg/s by the user in this case. An example is if you want to distribute the source over a fraction of the water depth, you could set z1=0.33 and z2=0.67 to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if z1 = 0.0 and z2 is 0.5, this suggests a vertical line source that is the lower one half of the water depth.

Area Sources

Area sources lie in a veritical or horizontal plane and are varying-release. Area sources are polygons and are specified by the locations of their vertices. The vertices must be ordered with a counter-clockwise convention. Particles are released from an area source such that there is a uniform distribution over the area and a Gaussian distribution perpendicular to the source. Source properties within an area source are uniform across the polygon.

Estimating generated Parcels

The 'Estimate Generate Parcels' dialog allows the user to edit the parcel mass of PTM sources (point, line, or polygon) to estimate the number of parcels created. This can be access by right-clicking on the PTM coverage and selecting 'Esitmate generated Parcels'.

Calculated vs. Specified Values

PTM can calculate the Fall Velocity, Critical Shear Initiation, and Critical Shear Deposition. Specifying a value of -1.0 for any of these parameters will result in the model calculating the value. See the model user manual for information on the equations used to calculate these values.

Related Topics

- PTM Source File
- PTM Traps

PTM Arc Attributes Dialog

Feature arcs created in a PTM type coverage can be used to create either sources or traps.

Related Topics

- PTM Sources
- PTM Traps

PTM Feature Point Attributes Dialog

The PTM feature point attributes dialog is used to specify point source attributes.

4.2.b. PTM Files

PTM Files

Input and Output files for PTM.

Input Files

- Required files
  - Program Control File (*.pcf) – run time instruction for the model.
  - Mesh file – bathymetry and boundary information for the model.
    - PTM Version 1.0 supports ADCIRC compatible three-noded triangular finite element mesh files (fort.14 or .grd). SMS contains tools to convert mesh files to this format.
  - Hydrodynamic input file – time varying water surface elevation and depth-averaged velocities.
    - The SMS 9.2 interface to PTM Version 1.0 supports XMDF (*.h5) format hydrodynamic files. SMS contains tools to convert mesh output files to this format.
  - Native sediment file – native sediments over the domain defined by the mesh file.
- Optional files
  - Sediment source file – the sediment sources in the simulation.
  - Trap file – locations where information about the simulation should be extracted.
  - Wave file(s) – time varying information about the wave field
  - Wave breaking file(s) – time varying information about wave breaking
  - PTM Boundary Condition file – describes the wet/dry interface

Output Files

The output files use the name given after the keyword :OUTPUT_PREFIX in the program control file and are appended with the following endings:

- **Mapping Output File** (*_maps.h5) – behavior of the flow and native sediments over the whole domain incremented by time.
- **Parcel Data File** (*_particles.h5) – parcel information incremented by time.
- **Echo file** (*_input.out) – ASCII file echoing the input parameters of the simulation.
- **Neighbor file** (*.neighbors) – contains mesh connectivity information. Once generated, can be specified as an input file to avoid regenerating for subsequent simulations with a given mesh. If the mesh changes, the neighbor file should be regenerated.
- **Trap Count File** (*_count.out) – contains the time and ID of particles entering a trap
- **Trap Residency File** (*_residency.out) – contains the time particles enter and exit a trap and the total time spent in the trap.

PTM Control File

The PTM program control file (*.pcf) contains run time instruction for the model.

File Overview

The program control file (PCF) contains all the information necessary for the PTM to perform the simulation requested. The PCF file is a keyword oriented file. Lines in the file should contain either comments, with the line beginning with a "#" symbol, or a keyword, a single word beginning with a colon, ":", and its associated values. Some keywords have no associated values, e.g.:

```
:MOBILITY_MAPPING
```

These are generally on/off instructions to the model. Keywords pointing ot filenames or setting constants have one or two associated values, e.g.:

```
:MESH_FORMAT ADCIRC
:MESH_FILE estuary.grd
```

Keywords and values are not case sensitive with the single exception of the simulation output file names (i.e. file names like "Run_C1_Tp_12s_paths.out" are possible).

The PCF file is read until the instruction :

```
:END_DATA
```

is encountered.

The following list summarizes the rules for construction of a PCF:

- Keywords should be preceeded with ":".
- Comments should be preceeded with "#".
- Keywords are not case sensitive.
- Keywords and value(s) are white space delimited (tab or space) and, so, should not contain blanks.
- Keywords may be placed in any order in the file.
- If a keyword appears more than once, then the final incidence is used.
- All value units are S.I.
- All times are relative to the clock defined by the currents file.

Example PCF File

```
##########################################################
# PTM Simulation file written by SMS 9.2.1
# SMS Build Date: Mar  9 2007
# Date: 03/09/07
# Time: 16:36:04
##########################################################
:CURRENTS
#
:ID_OUTPUT
:ELEVATION_OUTPUT
:GRAIN_SIZE_OUTPUT
:MOBILITY_OUTPUT
```

```
:STATE_OUTPUT
:PARCEL_MASS_OUTPUT
:OUTPUT_INC 10
:MAPPING_INC 1000
#
:BY_WEIGHT
:GRID_UPDATE 300
#
:MESH_FORMAT ADCIRC
:MESH_FILE estuary.grd
:FLOW_FORMAT XMDF
:FLOW_FILE_XMDF estuary_tide.h5
:XMDF_VEL_PATH Datasets/tide only velocity
:XMDF_WSE_PATH Datasets/tide only wse
:NEIGHBOR_FILE estuary.neighbors
:SOURCE_FILE estuary_Tides.source
:SEDIMENT_FILE estuary.sediments
:OUTPUT_PREFIX estuary_Tides
#
:START_RUN 2004 2 11 0 0 0
:STOP_RUN 2004 2 12 7 45 0
:START_FLOW 2004 2 9 0 0 0
#
:TIME_STEP 3.000000
#
:KET 0.25
:KEV 0.00859
:TEMPERATURE 15
:SALINITY 34
:RHOS 2650
:MIN_DEPTH 0.01
#
:ETMIN 0.02
:EVMIN 0
#
:EMRL_GUID_COV 7d0bcdf2-f218-4046-ba24-218294d74794
:EMRL_PARTICLE_SET_GUID b7183313-bcba-45d9-8cf7-f6ce96cf1d45
:END_DATA
```

Keywords

Keyword	Value(s)	Action
:waves		Instructs the model that waves are to be runs in the model. Turns on the various wave flags. Default is inactive.
:currents		Instructs the model that currents are to be runs in the model. Turns on the various wave flags. Default is inactive.
:no_parcels		Instructs the model to only perform Eulerian calculations (i.e. no particles are released). Source file is not required if this option is used. Default is inactive.
:ensim		Tells the model that the input bathymetry file uses the CHC EnSim convention of positive upwards. Default is inactive. Default is inactive.
:paths		Instructs the model to output the parcel path files. Default is inactive.
:morphology		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":bed_level_mapping ". Default is inactive.
:flow_mapping		Instructs the model to output the flow conditions on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:bedform_mapping		Instructs the model to output the predicted native sediment bedforms on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:wave_mapping		Instructs the model to output the three predicted wave parameters (H, T, θ) on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:mobility_mapping		Instructs the model to output the predicted mobility of the native sediments on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":morphology". Default is inactive.
:bed_level_mapping		Instructs the model to output the predicted bed evolution on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Same result as ":morphology". Default is inactive.
:bed_level_change_mapping		Instructs the model to output the predicted bed level change on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:transport_mapping		Instructs the model to output the predicted potential sediment transport rate of the native sediments on the Eulerian mesh node locations. Frequency of output is given by 'mapping_inc'. Default is inactive.
:grain_size_output		Instructs the model to output the grain size of the parcels. Frequency of output is given by 'output_inc'. Default is inactive.
:mobility_output		Instructs the model to output the mobility of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:elevation_output		Instructs the model to output the elevation of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:height_output		Instructs the model to output the height of each parcel above the bed. Frequency of output is given by 'output_inc'. Default is inactive.
:state_output		Instructs the model to output the state of each parcel (active=1, dormant=0). Frequency of output is given by 'output_inc'. Default is inactive.
:id_output		Instructs the model to output the identification number of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:source_output		Instructs the model to output the original source of the each parcel. Frequency of output is given by 'output_inc'. Default is inactive.
:flow_output		Instructs the model to output the velocity components of each parcel. Frequency of output is given by 'output_inc'. Default is inactive.

:density_output		Instructs the model to output the density of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:parcel_mass_output		Instructs the model to output the mass of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:time_step	<real>	The time step of the simulation in seconds, e.g.: • time_step 2. • Input value required.
:by_weight		Instructs the model to build up the distributions of parcel grain sizes at each source such that they are Gaussian in terms of weight (i.e. “percentage finer by weight”), rather than by grain size. Default is inactive.
:no_bedforms		Instructs the model to skip bedform calculations on the Eulerian grid. Default is inactive.
:bed_porosity		Input statement for bed porosity. Default value is 0.4.
:eulerian_method	ptm or van_rijn	Selects which method to use to determine the native sediment mobility and shears on the Eulerian grid. The two choices are ptm and van_rijn, e.g.: • eulerian_method van_rijn • The ptm method is much faster. Default is ptm.
:centroid_method	rouse or van_rijn	Selects which method to use to determine the parcel mobility and shears. The two choices are rouse and van_rijn, e.g.: • lagrangian_method rouse • The rouse method uses empirical approximations of integrated Rouse curves. The van_rijn method uses a numerical integration of the total sediment transport at the location of the parcel to compute the centroid elevation. The rouse method if much faster. Default is rouse.
:advection_method	1d (formerly 2d – not available in SMS interface) or 2d (formerly q3d - default) or 3d	Selects which method to use for the advection of the parcels. The three choices are 1d, 2d and 3d, e.g.: • advection_method 2d • The 1d method places the parcel at the elevation of the centroid of the local total load distribution. The 2d method allows the parcel to be above or below the total load centroid elevation with the parcel moving vertically towards the centroid. The 3d option allows the parcel to move freely in the vertical in response to a vertical force balance. Default is 2d.
:velocity_method	2d (log) or 2d (uniform) or 2d (two-point) or 3d or 3ds or 3dz	Selects which method to use to compute the velocity profile in the vertical, e.g.: • velocity_method 3dz • The 2d (uniform) option causes the model to use the given velocity throughout the water column. This method is designed primarily for testing. The 2d (log) option causes the model to use a logarithmic velocity profile. The 3d variants force the model to use the given three-dimensional velocity components. These should only be used with three-dimensional input. Default is 2d (log)
:mesh_format	ADCIRC	Gives the format of the mesh file, e.g.: • mesh_format ADCIRC • Options are CMS-2D, CMS-3D, ADCIRC, M2D and CH3D Default value is ADCIRC.

:mesh_file	<character>	Gives the name of the mesh file, e.g.: • mesh_file test.grd • Input value required.
:flow_format	ADCIRC or CMS-2D or CMS-3D or CH3D	Gives the format of the hydrodynamic files: • flow_format ADCIRC • Options are ADCIRC, CMS-2D (for M2D), CMS-3D (for M3D) and CH3D. Default value is ADCIRC.
:flow_files	<character> <character>	Gives names of the hydrodynamic files, e.g.: • flow_files A1.63 A1.64 • Note that the extensions are optional; if a '.' is not found in the filename, the *.63 and *.64 will be appended. Input values required.
:flow_file_uv	<character>	Gives names of the hydrodynamic flow file, e.g.: • flow_file_uv A1.64 • Note that the extensions are optional; if a '.' is not found in the filename, the *.64 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_z	<character>	Gives names of the hydrodynamic elevation file, e.g.: • flow_file_z A1.63 • Note that the extensions are optional; if a '.' is not found in the filename, the *.63 will be appended. Input values required. This command allows names with spaces, including directories.
:flow_file_xmdf	<character>	Gives the name of the XMDF format flow file. This is a single file in h5 format.
:flow_file_dst	<character>	Gives name of the density/salinity/temperature file, e.g.: • flow_file_dst A1.45 • This is only used for some ADCIRC-3D model input. If IDEN=0, then no file is required.
:bc_file	<character>	Gives the name of the boundary condition file. Format depends on model type.
:xmdf_wse_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Water Surface Elevation (63)'
:xmdf_vel_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Velocity (64)'
:xmdf_u_path	<character>	Gives the internal data structure path for the U velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_v_path	<character>	Gives the internal data structure path for the V velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:xmdf_w_path	<character>	Gives the internal data structure path for the W velocity component in the XMDF format flow file (M3D uses three scalars for the three components of velocity). The default is 'Datasets/U'
:m2d_control_file	<character>	Gives the name of the M2D control file {*.m2c}. This is only used for M2D version 3.02 and requires the mesh format to be specified as 'm2d', rather than 'cms-2d'
:xmdf_dep_path	<character>	Gives the internal data structure path for the depth in the XMDF format file (M3D only). The default is 'Datasets/Depth'

:xmdf_sal_path	<character>	Gives the internal data structure path for the salinity in the XMDF format file (M3D and CH3D). The default is 'Datasets/Salinity'
:xmdf_temp_path	<character>	Gives the internal data structure path for the temperature in the XMDF format file (M3D and CH3D). The default is 'Datasets/Temperature'
:xmdf_M3D_path	<character>	Gives the internal data structure path for the 3D data in the XMDF format file (M3D only). The default is blank.
:xmdf_d35_path	<character>	Gives the internal data structure path for the D35 values in the XMDF format sediment file. The default is blank.
:xmdf_d50_path	<character>	Gives the internal data structure path for the D50 values in the XMDF format sediment file. The default is blank.
:xmdf_d90_path	<character>	Gives the internal data structure path for the D90 values in the XMDF format sediment file. The default is blank.
:start_flow	yyyy mm dd hh mm ss	Time origin of the ADCIRC flow files, e.g.: • start_flow 1999 12 3 22 30 0 • Each output step of the ADCIRC .63 and .64 files has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_format	STWAVE or WABED or XMDF	Gives the format of the wave file, e.g.: • wave_format STWAVE • Options are STWAVE, WABED or XMDF. Default value is STWAVE.
:wave_files	<integer>	Gives the number of STWAVE .wav files that are to be used in the present simulation, e.g.: • wave_files 100 • The list of file names should start on the line immediately after the :end_data statement. Input value required if waves are used.
:wave_file_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: • wave_file_xmdf south_waves_example.h5 • This is the main (parent) grid. Input value required.
:wave_file_2_xmdf	<character>	Gives the name of the XMDF input file with wave data, e.g.: • wave_file_xmdf south_waves_example.h5 • This is the nested (child) grid. Input value required.
:wave_frames	<integer>	Gives the number of output frames (steps) that appear in each STWAVE .wav file, e.g.: • wave_frames 1 • In general, STWAVE files will contain a single frame if produced using the SMS steering module and more than one frame if produced using a stand-alone version of STWAVE (e.g. steering module run -> 100 files with one frame each; stand-alone -> 1 file with 100 frames. Input value required if waves are used.
:wave_step	<real>	Gives the time (in seconds) between each output frame (step) in the STWAVE .wav file(s), e.g.: • wave_step 7200. • Input value required if waves are used.
:m2d_step	<real>	Gives the time (in seconds) between each output frame (step) in the XMDF-format M2D file, e.g.: • m2d_step 3600. • New XMDF files have the time units specified, so this may now longer be required.

:wave_grid_angle	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin	<real>	x-ordinate of the STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin	<real>	y-ordinate of the STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:nested		Instructs the model that nested STWAVE grids are to be used. The list of file names should start on the line immediately after the list of parent STWAVE file names. The nested and parent STWAVE grids must have the same number of frames and the same times associated with each frame.
:wave_grid_angle_2	<real>	Gives the orientation angle (in degrees, clockwise from y-axis to N) of the nested STWAVE grid, e.g.: • wave_grid_angle 20. • Input value required if waves are used.
:wave_x_origin_2	<real>	x-ordinate of the nested STWAVE grid origin, e.g.: • wave_x_origin 1226722.50 • Input value required if waves are used.
:wave_y_origin_2	<real>	y-ordinate of the nested STWAVE grid origin, e.g.: • wave_y_origin 39990.15 • Input value required if waves are used.
:start_waves	yyyy mm dd hh mm ss	Time of the first STWAVE frame, e.g.: • start_waves 2004 12 3 0 30 0 • Input values required if waves are used.
:exceed_waves		Instructs model to hold waves steady, if simulation exceeds duration of wave input file
:source_file	<character>	Gives the name of the file containing the sediment source information, e.g.: • source_file test_2.source • Input value required.
:neighbor_file	<character>	Gives the name of the file containing the neighbor source information, e.g.: • neighbor_file test.neighbor • The model will generate a file if one is not supplied. Input value required.
:output_prefix	<character>	Gives the prefix of the files used for output, e.g.: • output_prefix west_test_1 • The model will append an extension depending on the nature of the output file. Input value required.
:undertow		The model computes wave undertow. Default is inactive. Needs to be made more general.
:trap_file	<character>	Gives the name of the trap file, e.g.: • trap_file trap_A.dat • See trap section for details. If this keyword command is not issued, then no trap is present. Default is no trap.
:xmdf_grid_path	<character>	Path to the main folder in an M2D h5 file. This is the folder with such data as the Origin, Bearing etc. The datasets folder is usually below this one.
:last_step_trap		Surveys all particles on the last time step and places them in a trap, whether they are moving or not.

:start_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, is operational, e.g.: • start_trap 2004 12 4 16 0 50 • Input values required if trap is used. Default start time is the start of the simulation.
:stop_trap	yyyy mm dd hh mm ss	Time at which the trap, if specified, ceases operation, e.g.: • stop_trap 2004 12 4 19 20 0 • Input values required if trap is used. Default stop time is the end of the simulation.
:sediment_format	ADCIRC or XMDF_PROPERTY or XMDF_DATASET or m2d	Gives the format of the native sediment file, e.g.: • sediment_format ADCIRC • Default value is ADCIRC.
:sediment_file	<character>	Gives the name of the file containing the grain sizes of the native sediments, e.g.: • sediment_file sed-A2.grd
:start_run	yyyy mm dd hh mm ss	Start time of the simulation, e.g.: • start_run 2004 11 4 16 0 0 • Input value required.
:stop_run	yyyy mm dd hh mm ss	Stop time of the simulation, e.g.: • stop_run 2004 11 14 13 30 0 • Must specify either :duration or :stop_run.
:duration	<real>	The duration of the simulation in seconds. Must specify either :duration or :stop_time.
:grid_update	<integer>	The increment in steps between updating of the mobility, shears and bedforms of the Eulerian grid
:flow_update	<Integer>	The increment in steps between updating of the flows and elevations of the Eulerian grid
:ket	<real>	Diffusion scalar. Default value is 0.25.
:temperature	<real>	The water temperature in °C. The default value is 15°C.
:salinity	<real>	The water salinity in ‰. The default value is 34 ‰.
:output_inc	<integer>	The increment in number of steps between output of the Lagrangian parcel data. The default value is 10.
:mapping_inc	<integer>	The increment in number of steps between output of the Eulerian grid data. The default value is 1000.
:min_depth	<real>	The minimum computation depth in m. The default value is 0.01 m.
:debug		Tells the model to turn off certain random generators routines and output stage times
:ensim_parcels		The model outputs a file with particle data in EnSim format (.pcl)
:ensim_maps		The model outputs a file with map data in EnSim format (.t3s)
:tecplot_parcels		The model outputs a file with particle data in Tecplot format (.plt)
:tecplot_maps		The model outputs a file with map data in Tecplot format (.plt)
:no_xmdf_parcels		The model will not output a parcel data file in XMDF format (.h5)
:no_xmdf_maps		The model will not output a map data file in XMDF format (.h5)
:population_record		A file with population history (e.g. particles born, trapped, deposited, etc.) is output as in Tecplot format as *_population.plt
:tau_cr_output		Instructs the model to output the critical shear for the initiation of motion of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.

:fall_velocity_output		Instructs the model to output the fall velocity of each parcel. Frequency of output is given by ‘output_inc’. Default is inactive.
:xmdf_compressed		Instructs the model to use compressed XMDF files. Default is inactive.
:no_hiding_exposure		Turns off hiding and exposure routines. Default is active.
:no_turbulent_shear		Turns off probabilistic shear calculations. Default is active.
:no_bed_interaction		Turns off bed interaction routines. Default is active.
:no_wave_mass_transport		Turns off wave mass transport calculations. Default is active.
:wave_mass_transport		Turns on wave mass transport calculations. Default is inactive.
:residence_calc		Model performs residency calculations (requires a polygon trap to be active)
:neutrally_buoyant		Model assumes all particles are neutrally buoyant (i.e. particle density set to that of the fluid and fall velocity is set to 0.). Default is inactive.
:bottom_flow_format	ADCIRC or XMDF	Gives the format of the bottom currents file. Options are ‘Default is ‘adcirc’.
:xmdf_bot_path	<character>	Gives the internal data structure path in the XMDF format flow file. The default is 'Datasets/Velocity (64)'
:bottom_flow_file	<character>	Gives the name of the bottom currents file.
:bottom_flow_height	<real>	Sets the height of the input bottom currents.
:bottom_mask_file	<character>	Gives the locations of the nodes where bottom currents are provided (1) and not provided (0)
:bottom_start_flow	yyyy mm dd hh mm ss	Time origin of the bottom flow files, e.g.: • bottom_start_flow 1997 10 5 12 45 0 • Each output step of the bottom flow file has a timestamp in seconds that is relative to a UTC time selected when the ADCIRC model control file is established. Input values required.
:wave_breaking		Instructs model to open a *.brk file for each *.wav file opened and to use breaking indices (Bi) estimate increased surf zone mixing.
:kew	<real>	Sets diffusion coefficient for waves. Default is 5. The horizontal eddy viscosity is adjusted by the factor:
:kev	<real>	Sets vertical diffusion coefficient. Default is 0.00859.
:etmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.02.
:evmin	<real>	Sets minimum vertical eddy viscosity. Default is 0.0.
:source_to_datum		Sets the z-value of parcels released by sources to a z-value relative to datum rather than the bed
:numerical_scheme	<Integer>	Controls the order of the numerical scheme. Only valid options are 2 or 4.
:rhos	<real>	Sets the density of the bed sediments. Default is 2650.
:emrl_particle_set_guid	<character>	Used by SMS. Model will pass string to output files.
:emrl*		Model will ignore lines with these first four characters, except :emrl_particle_set_guid.
:end_data		Indicates the end of the keyword part of the file. The only contents of the file beyond this point should be the names of the wave files (if required).

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Sediment File
- Source File
- Trap File

PTM Sediment File

The PTM native sediment file contains the grain size information for the native sediments in terms of its D35, D50, and D90.

File Overview

The native sediments file contains the spatially-varying grain size information for the native sediments in terms of D35, D50 , and D90 . Frictional characteristics of the bed are computed with D90 . The D50 value is used in the prediction of bed forms, in the determination of sediment mixing routines that influence reentrainment of deposited particles, and in the hiding and exposure routines that influence the critical shear stress of deposited particles. The D35 value is used in the determination of the suspended sediment transport if the van Rijn approach is selected for the centroid method. Non-erodible areas (e.g., rock outcroppings) can be identified by a negative input grain size. This absolute value of the grain size is treated as an effective roughness height, ks′ and ks′′.

The native sediments filename is specified on the Files page of the PTM model control. A native sediments file can be generated by the SMS interface, if one is required, by pressing the "Create input file9s) from data..." button on the Files tab of the PTM model control. This will open the Create PTM External Input Files dialog.

The native sediment file can be in ASCII or XMDF format.

ASCII File Overview

An ASCII format native sediment file is used when using an ADCIRC (fort.14 or *.grd), or CMS-Flow format mesh file (see PCF File keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1. The first line is a comment line that is not read by PTM.
2. The second line contains the number of elements and the number of nodes of the mesh.
3. The next section contains a single line for each node, which gives the node number, easting (m), northing (m), D35 (mm), D50 (mm), and D90 (mm).
4. The file extension should be specified as .sediments (for SMS)

Example ASCII Native Sediment Data File

```
Node ID   X(m)      Y(m)      D35(mm)   D50(mm)   D90(mm)
2704    1401
1         54010.00  60160.00  0.13      0.19      0.3
2         52008.90  60146.50  0.13      0.19      0.3
3         50007.80  60133.00  0.13      0.19      0.3
4         48006.70  60119.40  0.13      0.19      0.3
5         46005.60  60105.90  0.13      0.19      0.3

[Continued...]
```

```
    1397      -6825.58  -408.412  0.13      0.19      0.3
    1398      -6876.47  -481.815  0.13      0.19      0.3
    1399      -6950.30  -414.314  0.13      0.19      0.3
    1400      -6693.29  -533.139  0.13      0.19      0.3
    1401      -6793.13  -526.903  0.13      0.19      0.3
```

In this example, there are two information lines (one comment line and one line with the number of elements and number of nodes) followed by 1,401 lines of data. Each node has sediment with a D35 of 0.13 mm, D50 of 0.19 mm, and D90 of 0.3 mm.

XMDF File Overview

An XMDF format native sediment file is used when using an XMDF format mesh file (see keywords :MESH_FORMAT and :SEDIMENT_FORMAT).

1. D35, D50, and D90 will be the same for every node when given as a property.
2. D35, D50, and D90 can be spatially varied when given as a dataset.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Source File
- Trap File

PTM Source File

The PTM sediment source file contains the time varying location and parameters assigned to a source.

File Overview

The first line contains the number of Instant Mass Sources, NIMS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Instant Mass Source(s)
```

This is followed by NIMS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. In this example, there is one instant mass source, known as “My Instant Mass Source”, that has an id of zero and two instructions:

```
0  2  My Instant Mass Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the mass of each parcel (kg), the horizontal radius of the source (m), the vertical radius of the source, the mass of the source (kg), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2004 10 6 12 0 0 4.76400000000000e+003 2.63800000000000e+003 8 2 1 1 180 2e-007 0.8 2650 -1 -1 -1
2004 10 7 20 0 0 4.76400000000000e+003 2.63800000000000e+003 8 2 1 1 180 2e-007 0.8 2650 -1 -1 -1
```

The next set of data contains information for Point Mass Rate Sources. This begins with the number of Point Mass Rate sources, NPMRS. This is a single integer value. Any comments can follow on the line, e.g.:

```
2 Point Mass Rate Source(s)
```

This is followed by NPMRS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label. e.g.:

```
0  4  My Mass Rate Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the point at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
2004 10 6 10 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2 0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2 0.05 2e-006 0.8 2650 -1 -1 -1
2004 10 6 11 0 1 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2 0    2e-006 0.8 2650 -1 -1 -1
2004 10 7 20 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2 0    2e-006 0.8 2650 -1 -1 -1
```

The next set of data contains information for Line Sources. This begines with the number of Line Sources, NLS. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Line Source(s)
```

This is followed by NLS blocks of data. Each block of data contains:

- An identification line giving the line source id number, the number of insturctions, NIS, and a label, e.g.:

```
    1   4   My Horizontal Line Source
```

- NIS lines of instructions giving the time to issue the instruction (year, month, day, hour, minute, second), the x, y, and z location of the first end point of the line at that time (m), the x, y, and z location of the second end point of the line at that time (m), the horizontal radius of the source (m), the vertical radius of the source, the mass rate of the source (kg/s/m), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
    2004 10 6 18 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
    2004 10 6 19 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
    2004 10 6 19 0 1 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 0        2.5e-008
0.8 2650 -1 -1 -1
    2004 10 7 20 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 0        2.5e-008
0.8 2650 -1 -1 -1
```

The next set of data contains information for Polygon Sources. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Sources, NPS. This is a single integer value. Comments can follow on the line, e.g.:

```
    1 Polygon Source(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the source id number, the number of instructions, NIS, and a label, e.g.:

```
    1 2 My Polygon Source
```

- NIS blocks of instructions. Each block of instructions contains:
  - A line with the time to issue the instruction (year, month, day, hour, minute, second). For each point in the polygon (1 point per line), the x, y, and z location of the point (m). The final line of the block of instructions contains the horizontal radius of the source (m), the vertical radius of the source (m), the mass rate of the source (kg/s/m^2), the grain size (m), the standard deviation of the sediment distribution (Phi-units), the density (kg/m^3), the fall velocity (-1 to have PTM compute) (m/sec), Tau critical initiation (-1 to have PTM compute) (N/m^2), and Tau critical deposition (-1 to have PTM compute) (N/m^2) e.g.;

```
    2000  1    1 8 0 0
     2451 4195 2
     2453 4560 2
     2183 4544 2
     2000 4342 2
     2261 4129 2
    1     0    0 .00001 0.0003 0.4 2650 -1 -1 -1
```

The following list summarizes the rules for construction of a sediment source file:

- Data files must contain an entry stating the number of each type of sediment source, even if one or more sediment source types are not used.
- All sediment sources are time varying and are accompanied by a list of instructions.
- Linear interpolation between instructions is used to obtain source characteristics at a given time. All characteristics are interpolated. A constant source can be defined by two instructions with all data, except the start and stop times constant.
- The final instruction must have a time later than the stop time of the model.

Example Source File

```
1 Instant Mass Source(s)
 3  2  My Instant Mass Source
  2004 10 6 12 0 0 4.76400000000000e+003 2.63800000000000e+003 8 2 1 1
180 2e-007 0.8 2650 -1 -1 -1
  2005 10 6 20 0 0 4.76400000000000e+003 2.63800000000000e+003 8 2 1 1
180 2e-007 0.8 2650 -1 -1 -1
2 Point Mass Rate Source(s)
 2  4  My Moving Point Mass Rate Source
  2004 10 6 16 0 0 4.09100000000000e+003 3.37500000000000e+003 8 2 3 3
0.05 5e-008 0.8 2650 -1 -1 -1
  2004 10 6 17 0 0 4.18600000000000e+003 1.90700000000000e+003 8 2 3 3
0.05 5e-008 0.8 2650 -1 -1 -1
  2004 10 6 17 1 0 4.09100000000000e+003 3.37500000000000e+003 8 2 3 3
0 5e-008 0.8 2650 -1 -1 -1
  2005 10 6 20 0 0 4.09100000000000e+003 3.37500000000000e+003 8 2 3 3
0 5e-008 0.8 2650 -1 -1 -1
 1  4  My Point Mass Rate Source
  2004 10 6 10 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2
 0.05 2e-006 0.8 2650 -1 -1 -1
  2004 10 6 11 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2
 0.05 2e-006 0.8 2650 -1 -1 -1
  2004 10 6 11 1 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2
 0 2e-006 0.8 2650 -1 -1 -1
  2005 10 6 20 0 0 4.74500000000000e+003 3.76200000000000e+003 10 2 2 2
 0 2e-006 0.8 2650 -1 -1 -1
1 Line Source(s)
 4  4  My Horizontal Line Source
  2004 10 6 18 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
  2004 10 6 19 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 2.2e-005 2.5e-008
0.8 2650 -1 -1 -1
  2004 10 6 19 1 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 0 2.5e-008 0.8
2650 -1 -1 -1
```

```
  2005 10 6 20 0 0 3.57500000000000e+003 3.84500000000000e+003 10
3.57500000000000e+003 1.57000000000000e+003 10 2 1 1 0 2.5e-008 0.8
2650 -1 -1 -1
1 Polygon Source(s)
5 12 2 My Polygon Source
2004 10 6 12 0 0
 4.25800000000000e+003 1.88400000000000e+003 0
 4.13900000000000e+003 2.04000000000000e+003 0
 4.14400000000000e+003 2.19100000000000e+003 0
 4.19600000000000e+003 2.28900000000000e+003 0
 4.37700000000000e+003 2.34100000000000e+003 0
 4.53300000000000e+003 2.31000000000000e+003 0
 4.59500000000000e+003 2.27900000000000e+003 0
 4.64700000000000e+003 2.17500000000000e+003 0
 4.64700000000000e+003 2.08200000000000e+003 0
 4.58500000000000e+003 1.96800000000000e+003 0
 4.50700000000000e+003 1.91600000000000e+003 0
 4.37700000000000e+003 1.90000000000000e+003 0
2 1 1 0.05 0.0001 0.8 2650 -1 -1 -1
2005 10 6 20 0 0
 4.25800000000000e+003 1.88400000000000e+003 0
 4.13900000000000e+003 2.04000000000000e+003 0
 4.14400000000000e+003 2.19100000000000e+003 0
 4.19600000000000e+003 2.28900000000000e+003 0
 4.37700000000000e+003 2.34100000000000e+003 0
 4.53300000000000e+003 2.31000000000000e+003 0
 4.59500000000000e+003 2.27900000000000e+003 0
 4.64700000000000e+003 2.17500000000000e+003 0
 4.64700000000000e+003 2.08200000000000e+003 0
 4.58500000000000e+003 1.96800000000000e+003 0
 4.50700000000000e+003 1.91600000000000e+003 0
 4.37700000000000e+003 1.90000000000000e+003 0
2 1 1 0.05 0.0001 0.8 2650 -1 -1 -1
*****************************************************************************************
* Guide:
*****************************************************************************************
Number of Instant Mass Source(s)
   time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), mass(kg), grain_size(m), standard_dev(Phi-units),
density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)
Number of Point Mass Rate Source(s)
   time(6), x_location, y_location, z_location, Pmass(kg), Hradius(m),
Vradius(m), rate(kg/s), grain_size(m), standard_dev(Phi-units),
density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),
```

```
TAU_critical_deposition(N/m^2 (-1 = compute)
Number of Line Source(s)
   time(6), x1_loc, y1_loc, z1_loc, x2_loc, y2_loc, z2_loc, Pmass(kg),
Hradius(m), Vradius(m), rate(kg/s/m), grain_size(m), phi_sd(Phi-units),
 density(kg/m^3), fall_velocity(m/s) (-1 = compute),
TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)
Number of Polygon Source(s)
   time(6)
   For each point (1 point per line): x_location, y_location,
z_location
   Pmass(kg), Hradius(m), Vradius(m), rate(kg/s/m^2), grain_size(m),
standard_dev(Phi-units), density(kg/m^3), fall_velocity(m/s) (-1 =
compute), TAU_critical_initiation(N/m^2) (-1 = compute),
TAU_critical_deposition(N/m^2 (-1 = compute)
*******************************************************************************************
* Source file written by SMS 10.0.0 Development
* SMS Build Date: Jun 20 2007
* Date saved: 06/20/07
* Time saved: 09:43:14
*******************************************************************************************
```

Vertical Line Source Datums

The source file format has been changed to allow the specification of vertical line source datums:

Old Format

```
# Line Source(s)
 LineSourceID  #Instructions   SourceName
```

New Format

```
# Line Source(s)
 LineSourceID  #Instructions   SourceName  SourceDatum
```

The values available for the SourceDatum field are:

- beddatum (default)
- surfacedatum
- depthdistributed

The Sourcedatum has a default value of beddatum. If no datum is given in the file, then it is assumed to use the bed as the datum. Therefore, PTM will still process source files in the old format correctly.

Relative to the bed

This is the default and is what was already in PTM. Given the following input from the source file:

```
Time     X1     Y1     Z1     X2     Y2     Z2     PMass     Hrad     Vrad     Rate     Additional Info
```

Relative to the water surface

If the SourceDatum is surfacedatum then the code takes Z1 and Z2 as relative to the water surface. The rate is given in kg/m/s by the user.

Depth-distributed

If the SourceDatum is depthdistributed then the code takes the Z1 and Z2 as percentages of the water depth. The rate is given in kg/s by the user in this case. An example is if you want to distribute the source over a fraction of the water depth, you could set z1=0.33 and z2=0.67 to introduce the source over the middle third of the water column. The depth-distributed source is taken as a percentage from the bed. So if z1 = 0 and z2 is 0.5, this suggests a segment that is the lower one half of the water depth.

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Trap File

PTM Wave File

The PTM wave input files contain the time-varying phase-averaged wave information for the simulation in terms of significant wave height, Hs, peak period, Tp, and direction.

File Overview

The wave input files must contain information for each node in the mesh input file. Wave input files are usually time-varying. The wave input file can be in ASCII or XMDF format.

ASCII File Overview

ASCII format wave input files are used when using STWAVE or CMS-Wave (WABED) results.

Example ASCII Wave File

TODO: When the STWAVE and CMS-Wave pages are completed we should like to their description of *.wav and *.brk files here.

XMDF File Overview

Example XMDF Wave File

TODO: Link to CMS-Wave file

PTM Trap File

A PTM trap file defines areas into which parcels enter and are counted. The traps can be open or closed traps. When a parcel enters an open trap, it is free to leave the trap. When a parcel enters a closed trap, it is not allowed to leave the trap. If the trap is a single trap, each parcel is only counted once per simulation.

File Overview

The first line contains the number of line traps, NLT. This is a single integer value. Any comments can follow on the line, e.g.:

```
1 Line Trap(s)
```

This is followed by NLT blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label. In this example, there is one line trap, known as “My Line Trap”, that has an id of two:

```
2 My Line Trap
```

- A single line giving the trap direction, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
0  3  15  0  1
```

- Two lines giving the x, y location of the line trap endpoints:

```
3.99000000000000e+001 1.07600000000000e+002
4.92000000000000e+001 1.78200000000000e+002
```

The next set of data contains information for Polygon Traps. The points must be ordered using a standard counter-clockwise convention. This begins with the number of Polygon Traps, NPT. This is a single integer value. Comments can follow on the line, e.g.:

```
1 Polygon Trap(s)
```

This is followed by NPS blocks of data. Each block of data contains:

- An identification line giving the trap id number, and a label, e.g.:

```
1 My Polygon Trap
```

- A single line giving the number of points in the trap, NP, trap bottom elevation, trap top elevation, open trap flag, and single count trap flag:

```
8 2 5 1 0
```

- For each point in the polygon (1 point per line), the point id, x, y location of the point:

```
    1 1.78600000000000e+001 2.39300000000000e+001
    2 3.54200000000000e+001 1.46700000000000e+001
    3 7.29400000000000e+001 2.04200000000000e+001
    4 7.91600000000000e+001 5.42600000000000e+001
    5 6.11200000000000e+001 7.88400000000000e+001
    6 2.93600000000000e+001 8.14000000000000e+001
    7 9.08000000000000e+000 6.55900000000000e+001
    8 3.97000000000000e+000 5.18600000000000e+001
```

Trap File Flag Values

Flag	Value(s)
Trap direction	-1 (decreasing x-coordinate), 0 (any direction), or 1 (increasing x-coordinate)
Open trap	0 (closed trap) or 1 (open trap)
Single count trap	0 (Count parcels every time they enter a trap) or 1 (Count parcels once per simulation)

If the keyword :LAST_STEP_TRAP is included in the Program Control File (PCF File), the traps will only be active on the last time step of the PTM simulation.

Example Trap File

```
1 Line Trap(s)
 0 My Line Trap
  0  2  12  1  0
   -5.28000000000000e+000 7.08600000000000e+001
   4.67600000000000e+001 8.57100000000000e+001
1 Polygon Trap(s)
 1 My  Polygon Trap
  6 5 20 0 1
   1 3.55800000000000e+001 1.56300000000000e+001
   2 6.54300000000000e+001 1.45100000000000e+001
   3 7.64500000000000e+001 2.90400000000000e+001
   4 5.92100000000000e+001 5.42600000000000e+001
   5 3.36700000000000e+001 5.18600000000000e+001
   6 2.18500000000000e+001 3.06300000000000e+001
*****************************************************************************************************************
* Guide:
*****************************************************************************************************************
Number of Line Traps(s)
For each Line Trap:
  Trap ID, Trap Name
   Trap Direction, Trap Bottom, Trap Top, Open Trap, Single Trap
    First node x1_location, y1_location
    Second node x2_location, y2_location
Number of Polygon Trap(s)
For each Polygon Trap:
```

```
  Trap ID, Trap Name
   Number of points in trap, Trap Bottom, Trap Top, Open Trap, Single Count Trap
    For each point: (1 point per line): x_location, y_location
*****************************************************************************************************
* Trap Keyword Values:
*****************************************************************************************************
Line Trap Direction:
  -1 (decreasing x-coordinate), 0 (any direction), or 1 (increasing x-coordinate)
  Default value is 0 (any direction)
Open Trap:
  0 (closed trap) or 1 (open trap)
  Default value is 1 (open trap)
Single Count Trap:
  0 (Count parcels every time they enter a trap) or 1 (Count parcels once per simulation)
  Default value is 0 (Count parcels every time they enter a trap)
*****************************************************************************************************
* Trap file written by SMS 10.1.0 Development Release
* SMS Build Date: Sep  9 2008
* Date saved: 09/09/08
* Time saved: 14:05:03
*****************************************************************************************************
```

Related Topics

- Particle Module
- Particle Tracking Model (PTM)
- Program Control File (PCF File)
- Sediment File
- Source File

PTM Trap Output

Trap output files are generated by PTM when one or more traps have been specified (this can be done by linking to an existing "*.trap" file or SMS will create this file from a PTM coverage containing trap objects.

A PTM trap file is a tabular ASCII file which contains information about when parcels entered specific traps in a PTM simulation. The first row of the file must be a header row defining the contents of the file. Each row after the header defines a single entry event of a parcel into a trap

File Overview

The first line of a standard trap output files contains the header defining the standard five columns of output, e.g.:

```
  STEP      DATE          TIME        PARTICLE     TRAP
```

The trap output file may also include additional columns which give extra information about the parcel, e.g.:

```
  STEP      DATE          TIME        PARTICLE     TRAP         MASS        MEDIAN_SIZE         SOURCE
```

The header line is followed by entry lines. Each line defines an entry event of a parcel into a trap:

```
STEP     DATE         TIME      PARTICLE  TRAP  MASS  MEDIAN_SIZE  SOURCE
1178  2001/01/01  02:38:10.000     53      1     2       0.1         2
1215  2001/01/01  02:41:15.000     47      3     3      0.05         1
1233  2001/01/01  02:42:45.000     99      1     3       0.2         2
.
.
```

When SMS is asked to read a trap output file, the file is identified by the extension "**_count.out". If the extension has been modified, SMS can be instructed to read this file using the file import wizard.

When SMS reads the standard trap output file, a time series curve for each trap represented in the file is created counting the number of parcels which entered the trap and logging the time (in hours from the first parcel) that the parcels entered. This is a "Count" time series curve. These time series curves are monotonically increasing.

If one or more "Value" column is specified when reading the file, SMS also creates one curve for each trap showing the accumulated value from this column. In the example above, the "MASS" column could be mapped as a value and the cumulative mass of each parcel would be summed as it enters the trap.

If one or more "Filter" column is specified, SMS asks for filter ranges associated with this quantity. For example, the user may wish to only count the parcels with a mean grain size below 0.15 mm. In this case, the "MEDIAN_SIZE" column above would be mapped as a "Filter" and a range of (0.00-0.15) specified in the filter range dialog that appears after the import wizard specification is complete. A parcel will be counted as in the range if it's value is greater than the minimum and less than or equal to the maximum). Ranges that are specified with the same minimum and maximum values are ignored. The values are always interpreted as doubles even when they represent an integer quantity such as "SOURCE". In this case, the ranges could be specified as (0.5-1.5) and (1.5-2.0) to filter out parcels from a specific source. SMS will create additional curves (both for count and cumulative value) for each trap for the parcels that fall into each filter range.

Therefore, the total number of curves created when reading a trap output file would be:

```
  Num_Curves = (1 + Num_Value_Columns) * (1 + Number_Filter_Ranges) * Number_Traps
```

These curves can be viewed using the Plot Wizard and selecting PTM Gage plots.

PTM Boundary Condition File

This file follows the ADCIRC fort.14 format. The only difference is that the header line will identify the file as a PTM_BC file with a version number. Here is an example:

```
PTM_BC CMS-PTM v2.1.025
```

When SMS reads the keyword (PTM_BC) in the header, it will then convert the file into a mif/mid file, and read it into the GIS module. Blue is used to indicate open boundaries, and brown for closed boundaries. The reason for reading the file into the PTM interface and not the ADCIRC is that some users may not have the ADCIRC interface enabled, or may have an existing mesh that they don't want to replace.

4.2.c. PTM Graphical Interface

PTM Graphical Interface

The SMS interface to PTM includes tools for creating input files as well as post-processing capabilities. The PTM interface includes the following components:

Model Control

The PTM Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, computation parameters, output options, and other global settings.

Map Module

PTM sources and traps are created in the Map Module as feature objects in a PTM type coverage.

Feature Points

- Instant Mass Source – Release at a single point in time (Ex. accidental vessel spill)
- Point Mass Rate Source – Release over a period of time (Ex. leaking pipeline)
- Vertical Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line (Ex. bucket dredge)

Feature Arcs

- Horizontal Line Source – Release with a uniform distribution along the line and a two-dimensional Gaussian distribution in the plane perpendicular to the line
- Horizontal Line Trap – Count particles crossing a line
- Vertical Area Source – Under development

Feature Polygons

- Horizontal Area Source – Release with a uniform distribution over the area and a Gaussian distribution perpendicular to the source
- Horizontal Area Trap – Count particles entering an area

For more information on sources, see PTM Sources. For more information on traps, see PTM Traps.

Running the Model

The PTM Files are written automatically with the SMS project file or can be saved separately using the *File* | *Save PTM* or *File* | *Save As* menu commands. See PTM Files for more information on the files used for the PTM run.

PTM can be launched from SMS using the *PTM* | *Run Model* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *PTM* | *Model Check* menu command.

PTM Menu

See PTM Menu for more information.

PTM Menu

The following menu commands are available in the PTM Menu:

Command	Functionality
New Simulation	Creates a new PTM simulation and adds it to the Project Explorer.
Model Check	Checks the active PTM simulation for common input errors.
Model Control	Opens the Model Control dialog (used to organize input files, specify model parameters, choose output options, etc.).
Run Model	Launches the PTM model using the active PTM simulation input files.

PTM Model Check

The model check will warn you if it suspects a potential error has been made in creating the input files for PTM and suggest how to resolve the issue. The issues shown in the model checker may or may not cause problems, it is left to the modeler to decide if an issue will affect the results of their simulation.

4.3. Generic Grid Model

5. Coastal Numeric Models

CSHORE

CSHORE is a transect morphology model which computes sediment transport along a specified profile/transect for a set of storm conditions.

Creating a Simulation with Transect Coverages

Create a new CSHORE Simulation by:

- Right click in a blank section of the project explorer.
- Then select "New Simulation→CSHORE".
- This brings up the coverage dialog. Select and name coverages you wish to be created (keep the transect coverage selected). Press Ok.
- This creates a simulation along with a transect coverage (roughness coverage would also be created if the option was on).

Working with Transects

- In the transect coverage, select the "Feature arc tool".
- Then click out an arc (this creates the transect).
- Then select the "Select Feature arc tool". Double clicking on the arc will display the attributes in a dialog.
- The attributes dialog contains an elevation curve and also a permeable structure curve if turned on.
- If the transect arc was created over a scatter set, the curves can automatically be created by right-clicking on the transect coverage, then selecting "Extract Elevations". Then select the curve to extract to elevations or permeable structure.

Working with Roughness

The roughness coverage allows you to create different roughness zones defined within polygon boundaries.

- In the roughness coverage, select the "Feature arc tool".
- Click out multiple arcs that form a closed region.
- Then select "Feature Objects>Build Polygons" from the menu.
- Switch to the "Select Polygon tool".
- Double clicking on the created polygon will bring up a dialog allowing the roughness to be specified.

Viewing Transects Profile Plot

Once curves have been established on the transects, you can view the transects profile plot. This is done by:

- Right-clicking on Simulation, then selecting "View transects profile".

Model Parameters

The model parameters can be viewed by:

- Right-clicking on Simulation, then selecting "Model Control".

Running the Model

Running the model is accomplished by:

- Right-clicking on Simulation, then selecting "Save, Export and Launch CSHORE".
- This writes out the necessary input files for CSHORE in a folder called CSHORE in the same directory as the project.
- The model will run, creating text file output of the CSHORE results.
- Upon exiting the model wrapper dialog, the CSHORE results will be converted into a different format for SMS.

Solution View

To view the Solution View:

- Click on the "Plot Wizard" located on the bottom toolbar.
- Select "CSHORE Solution View".
- Click "Next".
- In the dialog there is a mode for "Profile" and "Time series" that can be chosen from the combo box on the top left side of the dialog. "Time series" may not be chosen if wave overtopping and/or moveable bottom are turned off for all of the simulations.
- In "Profile" mode, the user may select multiple simulations and datasets, along with a single transect and timestep.
- In "Time series" mode, the user may select multiple simulations, datasets, and transects.
- After the options are chosen, the user can hit "Update" in order to see a plot of the data on the right side of the dialog.

CSTORM-MS

Create HPC scripts

CSTORM-MS is a set of models organized into a system that work together to create models relative to coastal storms. Many of the models in the system are very compute intensive models and may run for long periods of time. Often it is desirable to run these models in a HPC (supercomputer) environment to take advantage of many processors. Some users may not be as familiar with the HPC environment or want a system to help regenerate scripts as needed. The 'Create HPC Scripts' tool within SMS helps users create scripts from parameters. The parameters may be changed and the scripts regenerated as necessary. The parameters can be saved with the project, exported/loaded into other projects, and stored as the default parameters for future CSTORM-MS simulations.

Platforms

Platforms are the term used for a collection of script parameters because often the scripts will be created to target a specific HPC platform. However, a user could create multiple sets of scripts for the same platform if there are different run parameters for different types of simulations. The user can create, delete, or copy the scripts from one platform to a new platform.

Programs

Each of the programs within the CSTORM-MS system has its own script within the platform. Each program has its own run requirements which are handled through SMS defined variables (see below).

Variables

Variables are used to modify the script text for export. Variables can make it easier to change a script by identifying the items that should be changed in a script and giving a description for the variable. When the script is generated, every instance of the variable within the script will be replaced with the text in the 'value' column. It is recommended but not required that variables use a naming convention similar to the SMS defined variables for easy identification.

SMS defined variables (program specific)

SMS defined variables provide information to scripts based upon simulation parameters. These parameters are specified elsewhere but may need to be incorporated into individual scripts. One example of this type of variable is the number of processors to use for ADCIRC. SMS knows how many processors ADCIRC wants to use and can automatically make this part of the script.

User defined variables (global)

User defined variables allow scripts to be generated generically and give descriptions for the script parameters needed. User defined variables are global (shared between all programs/platforms). If you wish to have a similar variable specified differently for separate programs, add the program name to your variables. For example, if PBL and ADCIRC both needed a walltime but it should be different use $PBL_WALLTIME and $ADCIRC_WALLTIME rather than $WALLTIME within your scripts.

5.1. ADCIRC

ADCIRC

ADCIRC	
Model Info	
Model type	Finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains.
Developer	Rick Luettich Joannes Westerink Randall Kolar Cline Dawson
Web site	http://www.adcirc.org
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • ADCIRC Several Sample problems can be found on the ADCIRC model developer's webpage [1]

The ADCIRC (Advanced Circulation) model is a finite element hydrodynamic model for coastal oceans, inlets, rivers and floodplains. The initial developers of the code were Rick Luettich (University of North Carolina at Chapel Hill) and Joannes Westerink (University of Notre Dame). Other principal developers include Randall Kolar (University of Oklahoma at Norman) and Cline Dawson (University of Texas at Austin). Various other groups are involved in development and support around the country.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the ADCIRC model. See ADCIRC Graphical Interface for more information.

The ADCIRC Graphical Interface contains tools to create and edit an ADCIRC simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to ADCIRC. If a mesh has already been created for a ADCIRC simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to ADCIRC. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the ADCIRC Menu. See ADCIRC Graphical Interface for more information.

Functionality

ADIRC is a system of computer programs for solving time dependent, free surface circulation and transport problems in two and three dimensions. These programs utilize the finite element method in space allowing the use of highly flexible, unstructured grids. Typical ADCIRC applications have included: (i) modeling tides and wind driven circulation, (ii) analysis of hurricane storm surge and flooding, (iii) dredging feasibility and material disposal studies, (iv) larval transport studies, (v) near shore marine operations.

For more information about the ADCIRC model visit www.adcirc.org [2].

Using the Model / Practical Notes

- There is an ADCIRC listserv that may be useful to keep up-to-date about the latest releases of ADCIRC and to post any questions about ADCIRC. It is adcirc@listserv.unc.edu [3]. If you would like to join please email Crystal Fulcher [4].

Related Topics

- SMS Models
- LTEA – Linear Truncation Error Analysis
- ADCIRC Database

External Links

- ADCIRC Home page [5]
- Mar 2002 ERDC/CHL CHETN-IV-40 Guidelines for Using Eastcoast 2001 Database of Tidal Constituents within Western North Atlantic Ocean, Gulf of Mexico and Caribbean Sea [36]
- Jun 2001 ERDC/CHL CHETN-IV-32 Leaky Internal-Barrier Normal-Flow Boundaries in the ADCIRC Coastal Hydrodynamics Code [39]
- Mar 2001 Technical Report CHL-98-32 Shinnecock Inlet, New York, site Investigation Report 4, Evaluation of Flood and Ebb shoal Sediment Source Alternatives for the West of Shinnecock Interim Project, New York [40]
- Dec 1999 Coastal Engineering Technical Note IV-21 Surface-Water Modeling System Tidal Constituents Toolbox for ADCIRC [41] [42]
- ADCIRC wiki hosted by Seahorse Coastal Consulting [6]
- Glacier Bay Test Case by Dave F. Hill [7]
- Assessment of ADCIRC's Wetting and Drying Algorithm [8]

References

[1] http://www.adcirc.org/Examples.html
[2] http://www.adcirc.org
[3] mailto:adcirc@listserv.unc.edu
[4] mailto:cfulcher@email.unc.edu
[5] http://adcirc.org/
[6] http://www.seahorsecoastal.com/wiki/doku.php
[7] http://water.engr.psu.edu/hill/research/glba/default.stm
[8] http://aos.princeton.edu/WWWPUBLIC/PROFS/adcirc_report.pdf

ADCIRC Database

In order to work with the ADCIRC Database, you will need to download the files separately. The needed files are:

1. Atlantic Database and Grid Files [1] – 74.9 MB
    - adcircnwattides.exe (utility)
    - ec2001.grd (adcirc grid file)
    - ec2001.tdb (adcirc harmonic output file)
    - tides.in (sample file)
    - tides_ec2001.f (source code)
3. Pacific Database and Grid Files [2] – 86.2 MB
    - adcircnepactides.exe (utility)
    - enpac2003.grd (adcirc grid file)
    - enpac2003.tdb (adcirc harmonic output file)
    - tides.in (sample file)
    - tides_enpac2003.f (source code)

These files should be unzipped and placed in C:\Program Files\SMS X.X\models (or a location of your choosing). You will then need to tell SMS where these files are located by going to Edit | Preferences | File Locations.

Related Topics

- ADCIRC

References

[1] http://sms.aquaveo.com/adcircnwattides.zip
[2] http://sms.aquaveo.com/adcircnepactides.zip

ADCIRC Files

Here are tables of all the availible Input and Output files with the ADCIRC model (All formats might not be supported by SMS)

- For more information on the formating of these files click the link on the file name and you will be redirected to the ADCIRC online documentation (ADCIRC.org [5])

Input Files

Name	Description	Required/Conditional
fort.14 [1]	Grid and Boundary Information File	Required
fort.15 [2]	Model Parameter and Periodic Boundary Condition File	Required
fort.10 [3]	Passive Scalar Transport Input File	Conditional
fort.11 [4]	Density Initial Condition Input File	Conditional
fort.13 [5]	Nodal Attributes File	Conditional
fort.19 [6]	Non-periodic Elevation Boundary Condition File	Conditional
fort.20 [7]	Non-periodic, Normal Flux Boundary Condition File	Conditional
fort.22 [8]	Single File Meteorological Forcing Input	Conditional
fort.200,... [9]	Multiple File Meterological Forcing Input	Conditional
fort.23 [10]	Wave Radiation Stress Forcing File	Conditional
fort.24 [11]	Self Attraction/Earth Load Tide Forcing File	Conditional
fort.67 or fort.68 [12]	2DDI Hot Start Files	Conditional

Output Files

Name	Description
fort.6 [13]	Screen Output
fort.16 [14]	General Diagnostic Output
fort.33 [15]	Iterative Solver ITPACKV 2D Diagnostic Output
fort.41 [16]	3D Density, Temperature and/or Salinity at Specified Recording Stations
fort.42 [17]	3D Velocity at Specified Recording Stations
fort.43 [18]	3D Turbulence at Specified Recording Stations
fort.44 [19]	3D Density, Temperature and/or Salinity at All Nodes in the Model Grid
fort.45 [20]	3D Velocity at All Nodes in the Model Grid
fort.46 [21]	3D Turbulence at All Nodes in the Model Grid
fort.51 [22]	Elevation Harmonic Constituents at Specified Elevation Recording Stations

fort.52 [23]	Depth-averaged Velocity Harmonic Constituents at Specified Velocity Recording Stations
fort.53 [24]	Elevation Harmonic Constituents at All Nodes in the Model Grid
fort.54 [25]	Depth-averaged Velocity Harmonic Constituents at All Nodes in the Model Grid
fort.55 [26]	Harmonic Constituent Diagnostic Output
fort.61 [27]	Elevation Time Series at Specified Elevation Recording Stations
fort.62 [28]	Depth-averaged Velocity Time Series at Specified Velocity Recording Stations
fort.63 [29]	Elevation Time Series at All Nodes in the Model Grid
fort.64 [30]	Depth-averaged Velocity Time Series at All Nodes in the Model Grid
fort.67, fort.68 [12]	Hot Start Output
fort.71 [31]	Atmospheric Pressure Time Series at Specified Meteorological Recording Stations
fort.72 [32]	Wind Velocity Time Series at Specified Meteorological Recording Stations
fort.73 [33]	Atmospheric Pressure Time Series at All Nodes in the Model Grid
fort.74 [34]	Wind Stress or Velocity Time Series at All Nodes in the Model Grid
fort.81 [35]	Depth-averaged Scalar Concentration Time Series at Specified Concentration Recording Stations
fort.83 [36]	Depth-averaged Scalar Concentration Time Series at All Nodes in the Model Grid
fort.91 [37]	Depth-averaged Density Fields at Specified Recording Stations
fort.93 [38]	Depth-averaged Density Fields at All Nodes in the Model Grid

References

[1] http://adcirc.org/documentv47/fort_14.html
[2] http://adcirc.org/documentv47/fort_15.html
[3] http://adcirc.org/documentv47/fort_10.html
[4] http://adcirc.org/documentv47/fort_11.html
[5] http://adcirc.org/documentv47/fort_13.html
[6] http://adcirc.org/documentv47/fort_19.html
[7] http://adcirc.org/documentv47/fort_20.html
[8] http://adcirc.org/documentv47/fort_22.html
[9] http://adcirc.org/documentv47/fort_200.html
[10] http://adcirc.org/documentv47/fort_23.html
[11] http://adcirc.org/documentv47/fort_24.html
[12] http://adcirc.org/documentv47/fort_6768.html
[13] http://adcirc.org/documentv47/fort_6.html
[14] http://adcirc.org/documentv47/fort_16.html
[15] http://adcirc.org/documentv47/fort_33.html
[16] http://adcirc.org/documentv47/fort_41.html
[17] http://adcirc.org/documentv47/fort_42.html
[18] http://adcirc.org/documentv47/fort_43.html
[19] http://adcirc.org/documentv47/fort_44.html
[20] http://adcirc.org/documentv47/fort_45.html
[21] http://adcirc.org/documentv47/fort_46.html
[22] http://adcirc.org/documentv47/fort_51.html
[23] http://adcirc.org/documentv47/fort_52.html
[24] http://adcirc.org/documentv47/fort_53.html

[25] http://adcirc.org/documentv47/fort_54.html
[26] http://adcirc.org/documentv47/fort_55.html
[27] http://adcirc.org/documentv47/fort_61.html
[28] http://adcirc.org/documentv47/fort_62.html
[29] http://adcirc.org/documentv47/fort_63.html
[30] http://adcirc.org/documentv47/fort_64.html
[31] http://adcirc.org/documentv47/fort_71.html
[32] http://adcirc.org/documentv47/fort_72.html
[33] http://adcirc.org/documentv47/fort_73.html
[34] http://adcirc.org/documentv47/fort_74.html
[35] http://adcirc.org/documentv47/fort_81.html
[36] http://adcirc.org/documentv47/fort_83.html
[37] http://adcirc.org/documentv47/fort_91.html
[38] http://adcirc.org/documentv47/fort_93.html

ADCIRC Graphical Interface

The ADCIRC Graphical Interface includes tools to assist with creating, editing and debugging an ADCIRC model. The ADCIRC interface exists in the 2D Mesh Module.

Model Control

The ADCIRC Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In ADCIRC boundary conditions are defined on nodestrings. The default boundary condition is a closed boundary (no flow). See ADCIRC BC Nodestrings for more information.

Running the Model

The ADCIRC files are written automatically with the SMS project file or can be saved separately using the *File* | *Save ADCIRC* or *File* | *Save As* menu commands. See ADCIRC Files for more information on the files used for the ADCIRC run.

ADCIRC can be launched from SMS using the *ADCIRC* | *Run ADCIRC* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *ADCIRC* | *Model Check* menu command.

ADCIRC Menu

See ADCIRC Menu for more information.

Related Topics

- Boundary Types
- Boundary Conditions – tidal forcing
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes

- Steering

ADCIRC Menu

The following menu commands are available in the ADCIRC Menu:

Boundary Conditions

- Assign BC (Boundary Condition)
- Define Nodal Attributes
- Assign Nodal Attributes
- Create Observation Station

Model Parameters

- Spatial Attributes
- Model Check
- Model Control
- Run ADCIRC

Mesh Generation

- Mesh Generation Toolbox
  - Localized Truncation Error Analysis (LTEA)

Related Topics

- ADCIRC Coverage
- ADCIRC Mesh
- Steering

ADCIRC Mesh

ADCIRC Meshes

ADCIRC computations are performed on a finite element mesh. ADCIRC meshes can only contain linear triangular elements. For more information about the mesh structure and using meshes see Mesh Module.

ADCIRC meshes are often generated using scalar density paving which helps define meshes that transition well between different sizes of elements (see Mesh Generation). Size functions used for scalar density paving can be defined using Linear Truncation Error Analysis (LTEA).

Example ADCIRC Mesh File (fort.14 or *.grd)

```
My Mesh File
200  125
        1    0.0000000000 5000.0000000000   10.0000000000
        2    0.0000000000 4500.0000000000   10.0000000000
        3    0.0000000000 4000.0000000000   10.0000000000
        4    0.0000000000 3500.0000000000   10.0000000000
        5    0.0000000000 3000.0000000000   10.0000000000
        6    0.0000000000 2500.0000000000   10.0000000000
        7    0.0000000000 2000.0000000000   10.0000000000
        8    0.0000000000 1500.0000000000   10.0000000000
        9    0.0000000000 1000.0000000000   10.0000000000
       10    0.0000000000  500.0000000000   10.0000000000
       11    0.0000000000    0.0000000000   10.0000000000
       12  387.2875168500 4679.5600328850   10.0000000000
       13  500.0000000000 5000.0000000000   10.0000000000
       14  448.0608478950 4243.2347889200   10.0000000000
       15  439.9566817750 3740.3946708250   10.0000000000
       16  438.0183235050 3237.7140888100   10.0000000000
       17  434.2179653200 2728.5012612700   10.0000000000
       18  434.4501409550 2223.5360522450   10.0000000000
       19  445.6728191600 1709.4083456900   10.0000000000
       20  441.7355135250 1217.0963461800   10.0000000000
       21  446.7690604800  665.5453978200   10.0000000000
       22  500.0000000000    0.0000000000   10.0000000000
       23  879.4691569000 4527.8977321650   10.0000000000
       24 1000.0000000000 5000.0000000000   10.0000000000
       25  915.9217524550 3970.1981044250   10.0000000000
       26  908.0623079800 3485.8982158350   10.0000000000
       27  865.4508483900 2975.6001854250   10.0000000000
       28  870.3584492700 2470.6287334900   10.0000000000
       29  865.3646748800 1963.5880264200   10.0000000000
       30  958.2489022100 1408.2747297650   10.0000000000
       31  846.2862793400  940.9107967100   10.0000000000
       32  894.5851194050  468.8869156450   10.0000000000
       33 1000.0000000000    0.0000000000   10.0000000000
```

```
34 1407.4389323650 4568.9272855350   10.0000000000
35 1319.0864158900 4152.9138196250   10.0000000000
36 1500.0000000000 5000.0000000000   10.0000000000
37 1519.2308151400 3630.3323211050   10.0000000000
38 1289.5949017700 3195.6810638750   10.0000000000
39 1309.3380986300 2727.7292707550   10.0000000000
40 1305.6561222300 2246.1068999650   10.0000000000
41 1287.1975618200 1772.0435687850   10.0000000000
42 1581.6880220050 1382.8502340350   10.0000000000
43 1352.9489540350  880.7930319200   10.0000000000
44 1338.3794076000  386.3892121500   10.0000000000
45 1500.0000000000    0.0000000000   10.0000000000
46 1912.7374514350 4563.7881224100   10.0000000000
47 1799.9414465350 4111.9865086650   10.0000000000
48 2000.0000000000 5000.0000000000   10.0000000000
49 2095.6095101550 3684.4249560900   10.0000000000
50 2029.7448911600 3256.1867318100   10.0000000000
51 1724.4261905750 3002.3231849500   10.0000000000
52 1757.6153478550 2485.5439440750   10.0000000000
53 1700.5538408750 1971.3218032750   10.0000000000
54 2022.3261809400 1718.2985960850   10.0000000000
55 2115.1877830250 1308.3558784300   10.0000000000
56 1908.2173673400  884.8414132300   10.0000000000
57 1796.4311279300  444.4909875000   10.0000000000
58 2000.0000000000    0.0000000000   10.0000000000
59 2421.8299088200 4565.2347923850   10.0000000000
60 2296.2815207500 4136.2010145200   10.0000000000
61 2500.0000000000 5000.0000000000   10.0000000000
62 2548.1854914450 3736.1664761600   10.0000000000
63 2245.7592831150 2715.5365617900   10.0000000000
64 2156.5989885850 2163.7822188800   10.0000000000
65 2576.1588612050 1255.8568827850   10.0000000000
66 2424.8132302500  862.3361492400   10.0000000000
67 2296.5804477550  435.0891291250   10.0000000000
68 2500.0000000000    0.0000000000   10.0000000000
69 2928.9231473700 4562.5355493550   10.0000000000
70 2813.0832006250 4131.9646648200   10.0000000000
71 3000.0000000000 5000.0000000000   10.0000000000
72 3005.2616900500 3650.6086203300   10.0000000000
73 3079.6575104700 3155.0929235250   10.0000000000
74 3026.1311773200 1347.8384132750   10.0000000000
75 3074.0212166200 1837.0105551000   10.0000000000
76 2941.1113740800  868.2621980900   10.0000000000
77 2803.0270930300  435.0218954200   10.0000000000
78 3000.0000000000    0.0000000000   10.0000000000
79 3428.1706765000 4553.5572992900   10.0000000000
80 3334.2421005900 4097.5833709050   10.0000000000
```

```
     81 3500.0000000000 5000.0000000000   10.0000000000
     82 3585.5001362100 3580.4710896600   10.0000000000
     83 3644.1599668700 3020.1398601150   10.0000000000
     84 3268.7966206450 2718.6831817150   10.0000000000
     85 3591.9118685300 1432.2295580150   10.0000000000
     86 3444.7797743450  901.6282802800   10.0000000000
     87 3639.1938445900 1996.7994288000   10.0000000000
     88 3191.2693888900 2293.1529447050   10.0000000000
     89 3312.8201976500  437.8216047050   10.0000000000
     90 3500.0000000000    0.0000000000   10.0000000000
     91 3875.4018613600 4597.6915792100   10.0000000000
     92 3897.1163185450 4126.3881442500   10.0000000000
     93 4000.0000000000 5000.0000000000   10.0000000000
     94 4132.8748876900 3728.5069540050   10.0000000000
     95 4108.4300319250 3272.5630823750   10.0000000000
     96 4124.3068688700 2760.4946533350   10.0000000000
     97 3697.0596781500 2499.0463538250   10.0000000000
     98 4103.2022573000 1742.3164195700   10.0000000000
     99 4136.5962459150 1283.2373595850   10.0000000000
    100 3961.1675874700  903.0618231000   10.0000000000
    101 3813.1576567050  450.2947123950   10.0000000000
    102 4125.8315088050 2248.5711365450   10.0000000000
    103 4000.0000000000    0.0000000000   10.0000000000
    104 4367.7984401650 4498.4664948950   10.0000000000
    105 4521.7565602000 4046.3826573000   10.0000000000
    106 4500.0000000000 5000.0000000000   10.0000000000
    107 4598.8223028500 3503.3617712150   10.0000000000
    108 4561.3391885400 2955.6940550700   10.0000000000
    109 4563.9298111000 2441.5637730600   10.0000000000
    110 4561.8541878800 1933.3440965250   10.0000000000
    111 4569.5749317300 1429.5434022850   10.0000000000
    112 4527.5800133650  930.2255863050   10.0000000000
    113 4350.9621391150  487.6105782500   10.0000000000
    114 4500.0000000000    0.0000000000   10.0000000000
    115 5000.0000000000 4500.0000000000   10.0000000000
    116 5000.0000000000 4000.0000000000   10.0000000000
    117 5000.0000000000 5000.0000000000   10.0000000000
    118 5000.0000000000 3500.0000000000   10.0000000000
    119 5000.0000000000 3000.0000000000   10.0000000000
    120 5000.0000000000 2500.0000000000   10.0000000000
    121 5000.0000000000 2000.0000000000   10.0000000000
    122 5000.0000000000 1500.0000000000   10.0000000000
    123 5000.0000000000 1000.0000000000   10.0000000000
    124 5000.0000000000  500.0000000000   10.0000000000
    125 5000.0000000000    0.0000000000   10.0000000000
1    3     1    12    13
2    3    12     1     2
```

```
  3    3    12     2    14
  4    3     2     3    14
  5    3    15    14     3
  6    3     3     4    15
  7    3     4    16    15
  8    3     4     5    16
  9    3    17    16     5
 10    3     5     6    17
 11    3    18    17     6
 12    3     6     7    18
 13    3    19    18     7
 14    3    19     7     8
 15    3     8    20    19
 16    3    20     8     9
 17    3     9    21    20
 18    3     9    10    21
 19    3    22    21    10
 20    3    10    11    22
 21    3    13    12    23
 22    3    23    12    14
 23    3    13    23    24
 24    3    14    25    23
 25    3    14    15    25
 26    3    25    15    26
 27    3    15    16    26
 28    3    26    16    27
 29    3    17    27    16
 30    3    17    28    27
 31    3    17    18    28
 32    3    28    18    29
 33    3    29    18    19
 34    3    19    30    29
 35    3    30    19    20
 36    3    20    31    30
 37    3    20    21    31
 38    3    31    21    32
 39    3    22    32    21
 40    3    32    22    33
 41    3    24    23    34
 42    3    23    35    34
 43    3    23    25    35
 44    3    36    24    34
 45    3    35    25    37
 46    3    25    26    37
 47    3    37    26    38
 48    3    27    38    26
 49    3    38    27    39
```

```
50    3    39    27    28
51    3    28    40    39
52    3    40    28    29
53    3    40    29    41
54    3    29    30    41
55    3    41    30    42
56    3    42    30    43
57    3    31    43    30
58    3    31    32    43
59    3    43    32    44
60    3    33    44    32
61    3    33    45    44
62    3    36    34    46
63    3    46    34    47
64    3    34    35    47
65    3    35    37    47
66    3    48    36    46
67    3    47    37    49
68    3    49    37    50
69    3    50    37    51
70    3    37    38    51
71    3    38    39    51
72    3    51    39    52
73    3    39    40    52
74    3    40    53    52
75    3    53    40    41
76    3    41    42    53
77    3    42    54    53
78    3    54    42    55
79    3    42    56    55
80    3    43    56    42
81    3    57    56    43
82    3    57    43    44
83    3    44    45    57
84    3    58    57    45
85    3    59    48    46
86    3    46    60    59
87    3    47    60    46
88    3    60    47    49
89    3    59    61    48
90    3    49    62    60
91    3    62    49    50
92    3    51    63    50
93    3    63    51    52
94    3    52    64    63
95    3    52    53    64
96    3    64    53    54
```

```
 97    3    65    54    55
 98    3    55    66    65
 99    3    56    66    55
100    3    56    67    66
101    3    56    57    67
102    3    58    67    57
103    3    67    58    68
104    3    61    59    69
105    3    70    69    59
106    3    59    60    70
107    3    60    62    70
108    3    69    71    61
109    3    70    62    72
110    3    72    62    73
111    3    74    75    65
112    3    65    76    74
113    3    76    65    66
114    3    76    66    77
115    3    67    77    66
116    3    77    67    68
117    3    77    68    78
118    3    79    71    69
119    3    69    80    79
120    3    70    80    69
121    3    70    72    80
122    3    81    71    79
123    3    80    72    82
124    3    72    73    82
125    3    82    73    83
126    3    73    84    83
127    3    75    74    85
128    3    85    74    86
129    3    76    86    74
130    3    88    75    87
131    3    87    75    85
132    3    76    89    86
133    3    76    77    89
134    3    78    89    77
135    3    78    90    89
136    3    79    91    81
137    3    79    92    91
138    3    92    79    80
139    3    80    82    92
140    3    91    93    81
141    3    94    92    82
142    3    94    82    95
143    3    95    82    83
```

```
144    3    95    83    96
145    3    83    97    96
146    3    83    84    97
147    3    84    88    97
148    3    87    85    98
149    3    98    85    99
150    3    85   100    99
151    3   100    85    86
152    3   100    86   101
153    3    86    89   101
154    3    97    88    87
155    3    97    87   102
156    3   102    87    98
157    3   101    89    90
158    3    90   103   101
159    3   104    93    91
160    3   104    91    92
161    3   104    92   105
162    3    92    94   105
163    3   106    93   104
164    3    94   107   105
165    3   107    94    95
166    3    95   108   107
167    3    96   108    95
168    3    96   109   108
169    3    96   102   109
170    3    97   102    96
171    3   110   102    98
172    3   110    98   111
173    3    98    99   111
174    3   111    99   112
175    3   100   112    99
176    3   100   113   112
177    3   100   101   113
178    3   113   101   103
179    3   102   110   109
180    3   103   114   113
181    3   104   115   106
182    3   115   104   105
183    3   105   116   115
184    3   116   105   107
185    3   117   106   115
186    3   116   107   118
187    3   118   107   119
188    3   119   107   108
189    3   119   108   120
190    3   120   108   109
```

```
 191    3   109   121   120
 192    3   109   110   121
 193    3   122   121   110
 194    3   110   111   122
 195    3   123   122   111
 196    3   111   112   123
 197    3   123   112   124
 198    3   124   112   113
 199    3   114   124   113
 200    3   124   114   125
1 = Number of open boundaries
11 = Total number of open boundary nodes
11 = Number of nodes for open boundary 1
1
2
3
4
5
6
7
8
9
10
11
4 = Number of land boundaries
44 = Total number of land boundary nodes
11 1 = Number of nodes for land boundary 1
50
62
73
84
88
75
65
54
64
63
50
11 22 = Number of nodes for land boundary 2
125
124
123
122
121
120
119
118
```

```
116
115
117
11 0 = Number of nodes for land boundary 3
11
22
33
45
58
68
78
90
103
114
125
11 0 = Number of nodes for land boundary 4
117
106
93
81
71
61
48
36
24
13
1
```

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Model Control
- Spatial Attributes
- Steering

ADCIRC Model Control

ADCIRC Model Control

The ADCIRC Model Control dialog is where important project parameters are chosen and defined. There are 6 different tabs the user should be aware of. They are listed below.

- **General –** contains such options as model type, cold/hot start, terms, maximum number of iterations, bottom stress/friction, etc.
- **Timing –** define simulation run time, start time, and time step options
- **Files –** specify which input and output files are desired for the project
- **Tidal/Harmonics –** define tidal constituents and harmonic analysis options
- **Wind** - enable specific wind types and their options
- **Sediment Options –** enable sediment transport options

General Tab

Here is a list of terms found on the General Tab:

- **Model Type –** 2D DI, 3D VS, 3D DSS – The two dimensional depth integrated is the only option that is recommended for commercial applications at this time.
- **Initial Values –** Cold Start, Hot Start 1, Hot Start2. This option allows you to choose a file to initialize the values of water level and velocity for a simulation. The files must have been saved from a previous run using the same finite element grid.
- **Coriolis Option –** Constant, Variable. The type of Coriolis forcing used does not impact run time for ADCIRC at all. Variable should be used when the coordinate system is geographic. In this case, ADCIRC will load a Coriolis coefficient for each node based on the latitude of the node. If you wish to run ADCIRC in rectilinear coordinates you should specify a constant Coriolis option because the model does not support reprojection from rectilinear to geographic to determine individual latitudes for each node.
- **Minimum Angle For Tangential Flow**
- **Terms –** Finite Amplitude Terms, Advective Terms, Time Derivative Terms
- **Solver Type –** Lumped, Direct Banded, Iterative JCG - The Iterative JCG is the default and recommended solver. The other options are in the interface for backward compatibility and analytic tests.
- **Absolute Convergence Criteria**
- **Maximum Number of Iterations per Time Step**
- **Generalized Properties –** Wave Continuity, Lateral Viscosity
- **Bottom Stress/Friction –** Constant Linear, Constant Quadratic, Constant Hybrid, Varying Linear, Varying Quadratic, Friction Coefficient – The "Constant Linear" option should only be used for analytical cases when you are verifying the code. Linear friction causes overdamping in deep water. For "Constant Quadratic" the value of TAU0 should be set based on the principle depth. A value of TAU0 = 0.005 is suggested for deeper water (greater than 10 m depth). For shallow water, TAU0 = 0.02 is recommended. If your domain includes both shallow and deep water, you may want to consider the varying quadratic option.

Timing Tab

Important parameters to be aware of:

- **Ramp Function Value** – this value is a period of time, starting at the beginning of a simulation, that allows the full forcing of the tidal constituents to be gradually applied to the model. This prevents any issues or negative effects from occurring that could harm the results of a simulation.[1]
- **Reference time** – You can use this to specify that the extracted tidal constituents are earlier or later than the start time of the simulation (see ADCIRC documentation Reftime [2]).
- **Start Day** – the start time of a project
- **Time Step** – time interval between successive measurements
- **Run Time** – total length of time a simulation is run
- **End Time** – day when simulation should finish

Files Tab

This tab contains two spreadsheets. The first of which is concerned with the input information ADCIRC can use before a simulation is run. The user may toggle on or off any of the enabled input files by checking the checkbox in the Input column. Some of the more common input files that are used are:

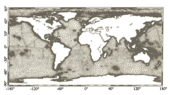
- **Initial Water Level** – provides initial water elevation levels
- **Hotstart 1 (fort.67)** – see ADCIRC Hot Start Files (fort.67 & fort.68) [3]
- **Hotstart 2 (fort.68)** – see ADCIRC Hot Start Files (fort.67 & fort.68) [3]

The second spreadsheet on this tab lists all of the possible output/solution files that ADCIRC can produce. This spreadsheet works similarly to the Input Files spreadsheet. A user may toggle on or off any of the output files listed by clicking in a checkbox in the Output column. Some of the files also require a start time, end time, or frequency to be specified for results to be produced. The start time and end time do not have to be the same as the start and end time of the entire simulation, but they do need to be somewhere within that time frame.

Tidal/Harmonics Tab

Tidal constituents are variations in tides that are created by different frequencies of astronomical forcing. They arise due to the gravitational influences of the Moon and Sun on the Earth, the tilt of the rotational axis of the Earth, the elliptical shape of the Moon's orbit around the Earth, the shape of the Earth's orbit around the Sun, and other such factors. Many constituents have been defined and are classified based on their cycle lengths. Most of the tidal constituents used in the ADCIRC interface of SMS are either diurnal (one cycle per day) or semidiurnal (two cycles per day) in nature. Harmonic constituents are those variations that have periods of less than half a day. For a table of commonly used constituents see Principal Tidal Constituents [42].[1]

SMS uses the following databases to provide the tidal constituent data to an ADCIRC simulation:

- LeProvost – The LeProvost database is a set of *.legi files which provide amplitude and phase information. The LeProvost database can be downloaded at: sms.aquaveo.com/leprovost.zip [4].
  - An image of the LeProvost tidal database domain coverage can be found here –  [5]
- ADCIRC – The ADCIRC database includes a grid file and a harmonics file that uses an .exe extractor to put information into SMS. The ADCIRC database can be downloaded at: sms.aquaveo.com/adcirctides.zip [6]. More information about the ADCIRC database can be obtained at the ADCIRC web page: ADCIRC Tides Databases [7].

The path to each database on your computer must be specified in the Preferences dialog by going to Edit|Preferences|File Locations.

Wind Tab

ADCIRC can model and compute wind velocities and stresses. The ADCIRC model is able to use wind data from a variety of different file types. The SMS interface is not able to read and create all of the available wind data file types. There are three types that SMS is able to read and create. Even though SMS does not read and create these other file types, it does tell ADCIRC that there is data there, where it is, and how to access it for a simulation to run correctly.

One of the parameters found in the fort.22 file (Single File Meteorological Forcing Input File [8]) will tell a user which of the wind data file types SMS can read and create. The first parameter specified in the fort.22 file is called the NWS value. This value represents what type of wind data is in the file. It can range from 0 to 100+. SMS can read and create fort.22 files that are of the NWS value of 1, 2, or 5. All other values of the NWS value are only shown and directed to the ADCIRC model by SMS so that a simulation will run correctly.

Sediment Options Tab

This section of the SMS interface is still under development and is not fully functional. It will be updated sometime in the future.

References

[1] Militello, A., and Zundel, A. K. (1999). “Surface-water modeling system tidal constituents toolbox for ADCIRC,” Coastal Engineering Technical Note CETN IV-21, U.S. Army Engineer Research and Development Center, Vicksburg, MS.
[2] http://adcirc.org/documentv47/parameter_defs.html#REFTIM
[3] http://www.adcirc.org/documentv46/fort_6768.html
[4] http://sms.aquaveo.com/leprovost.zip
[5] Le Provost, C., Genco, M. L., and Lyard, F. (1995). “Modeling and predicting tides over the World Ocean,” Quantitative Skill Assessment for Coastal Ocean Models, Coastal and Estuaring Studies, Vol. 47, pp 175-201.
[6] http://sms.aquaveo.com/adcirctides.zip
[7] http://www.unc.edu/ims/ccats/tides/tides.htm
[8] http://www.adcirc.org/documentv46/fort_22.html

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Spatial Attributes
- Steering

ADCIRC Point Attributes Dialog

The Refine Attributes dialog is used to set the attributes for a refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each refine point include:

- Refine point (checked = on)
  - Element size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

Related Topics

- Feature Objects Menu
- Mesh Generation

ADCIRC Spatial Attributes

ADCIRC Spatial Attributes

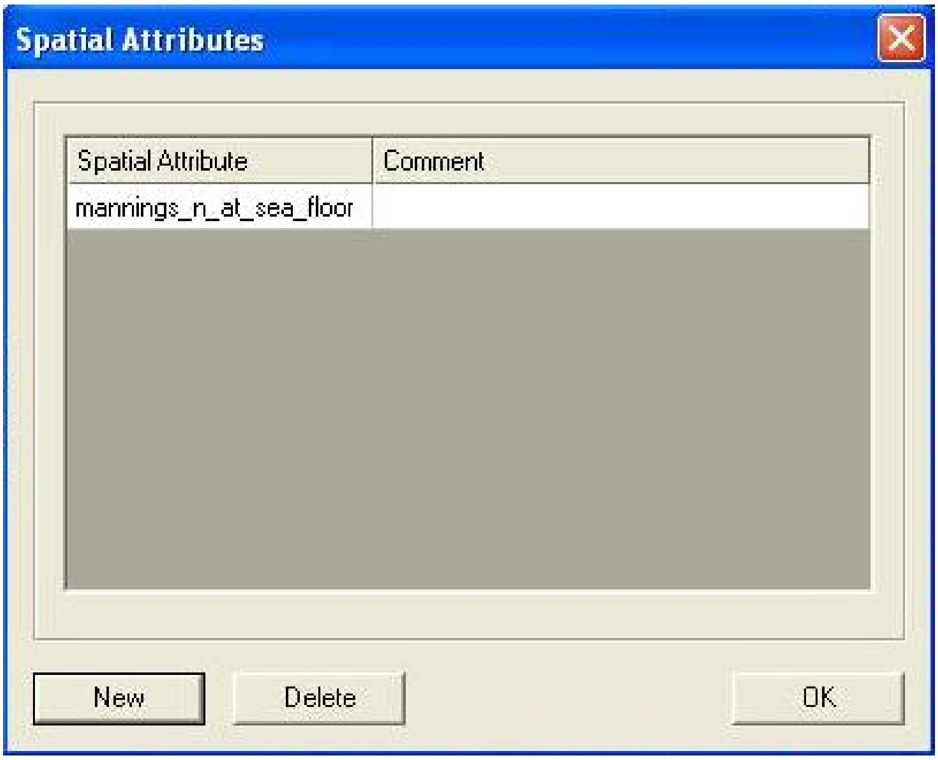
Spatial attributes are parameters that can be applied to the entire space of the simulation across the nodes of the mesh. ADCIRC is capable of handling such attributes in connection with the SMS interface. The user can define these attributes in a the *Spatial Attributes* dialog under the *ADCIRC* menu. There are a number of predefined attributes that the user can use or they can define their own. Here is a list of the predefined spatial attributes and a brief meaning of each.

- primitive weighting in continuity equation – Tau0
- surface submergence state – StartDry
- quadratic friction coefficient at sea floor – Fric
- surface directional effective roughness length – z0Land
- surface canopy coefficient – VCanopy
- bridge pilings friction parameters – BK, BAlpha, BDelX, POAN
- Manning's n at sea floor – ManningsN
- Chezy friction coefficient at sea floor – ChezyFric
- sea surface height above geoid – GeoidOffset
- horizontal eddy viscosity – ESLM

Please visit the ADCIRC web site [1] to learn more about the meaning of each of the spatial attributes listed above.

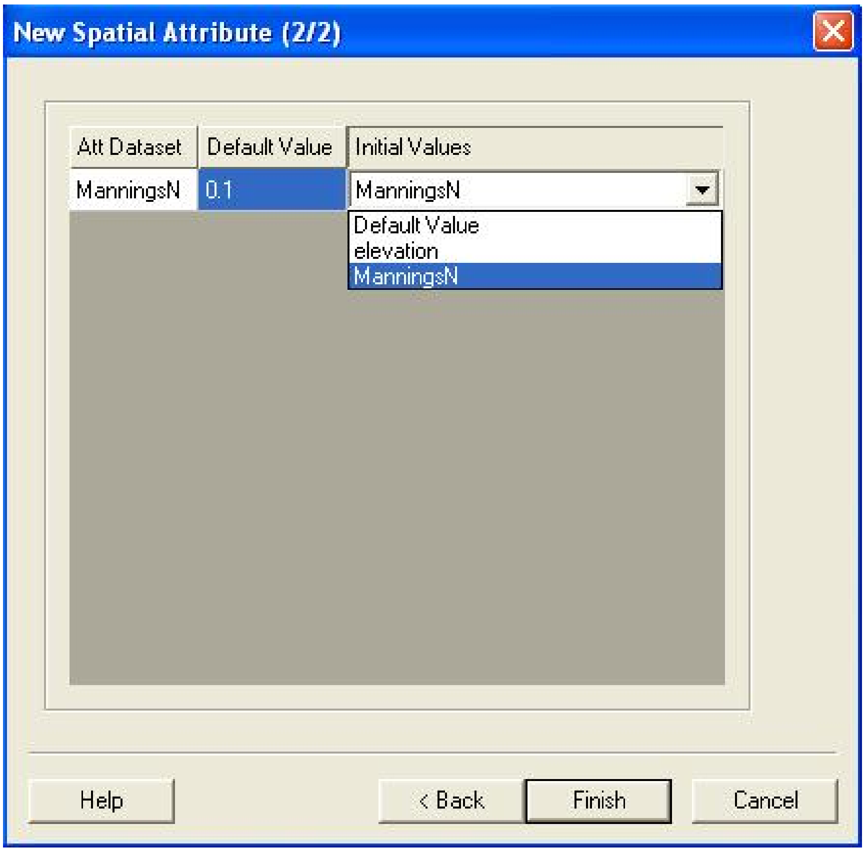
How to Use Spatial Attributes in a Simulation

Spatial Attributes are very easy to define and implement into an ADCIRC simulation. First, open the Spatial Attributes dialog from the ADCIRC menu. A dialog with a spreadsheet will appear that shows a list of the names and a comment of each currently defined spatial attribute. From here, you can create a new attribute or delete an existing attribute.



Creating a New Spatial Attribute

1. Click the New button in the Spatial Attributes dialog.
2. In the New Spatial Attribute dialog, select between the Predefined or Custom options. The Custom option will be used mostly for ADCIRC developers only. If Predefined is selected, select an attribute from the list-box and add a comment if so desired. If Custom is selected, type in a name for the new attribute and a comment, and then type the dataset name(s) for that attribute into the spreadsheet. (Before you can define a custom attribute, you must first read in a dataset of values into SMS.) Click the Next button to go to the next page of the dialog.
3. Notice, in the spreadsheet, all of the defined spatial attributes. The first column of the spreadsheet is called *Att Dataset*, which tells you the title of the attribute dataset. The next column is entitled *Default Value*. In this column you can define a constant value for the attribute. The last column of the spreadsheet is entitled *Initial Values*. Here you can choose which dataset the values of the attribute come from. Set the value of the attribute by entering a value in the *Default Value* column or pick a dataset for it from the *Initial Values* column. Click Finish to finalize the definition of the attribute.
4. Click OK to exit the Spatial Attributes dialog. Notice that the attribute just defined appears in the Project Explorer under the Mesh Data folder. Its dataset can be selected and contoured for viewing.



Editing Spatial Attributes

To edit a spatial attribute, open the Spatial Attributes dialog. Here you can create a new attribute to replace the one you want to edit. Once you have redefined the new attribute, SMS will ask you if you want to replace the existing one. Click OK to replace the old attribute with the new one that was just defined.

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Steering

References

[1] http://www.adcirc.org/documentv46/fort_13.html

ADCIRC Spatial Attributes 11.1

ADCIRC Spatial Attributes

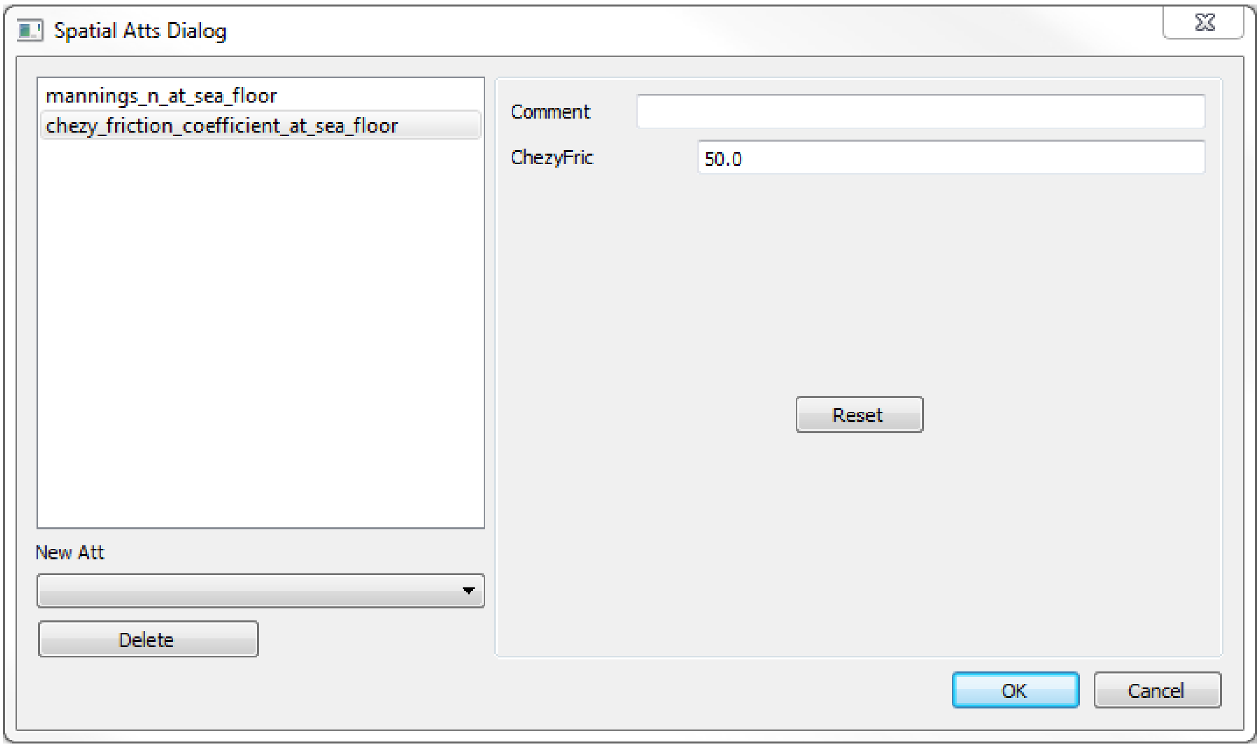
Spatial attributes are parameters that can be applied to the entire space of the simulation across the nodes of the mesh. ADCIRC is capable of handling such attributes in connection with the SMS interface. The user can define these attributes in a the *Spatial Attributes* dialog under the *ADCIRC* menu. There are a number of predefined attributes that the user can use or they can define their own. Here is a list of the predefined spatial attributes and a brief meaning of each.

- primitive weighting in continuity equation – Tau0
- surface submergence state – StartDry
- quadratic friction coefficient at sea floor – Fric
- surface directional effective roughness length – z0Land
- surface canopy coefficient – VCanopy
- bridge pilings friction parameters – BK, BAlpha, BDelX, POAN
- Manning's n at sea floor – ManningsN
- Chezy friction coefficient at sea floor – ChezyFric
- sea surface height above geoid – GeoidOffset
- horizontal eddy viscosity – ESLM

Please visit the ADCIRC web site [1] to learn more about the meaning of each of the spatial attributes listed above.

How to Use Spatial Attributes in a Simulation

Spatial attributes are managed using editable datasets within SMS. The creation and management of spatial attributes is managed through the Spatial Attributes Dialog. You can access the Spatial Attributes Dialog by selecting "ADCIRC | Spatial Attributes" from the menu.



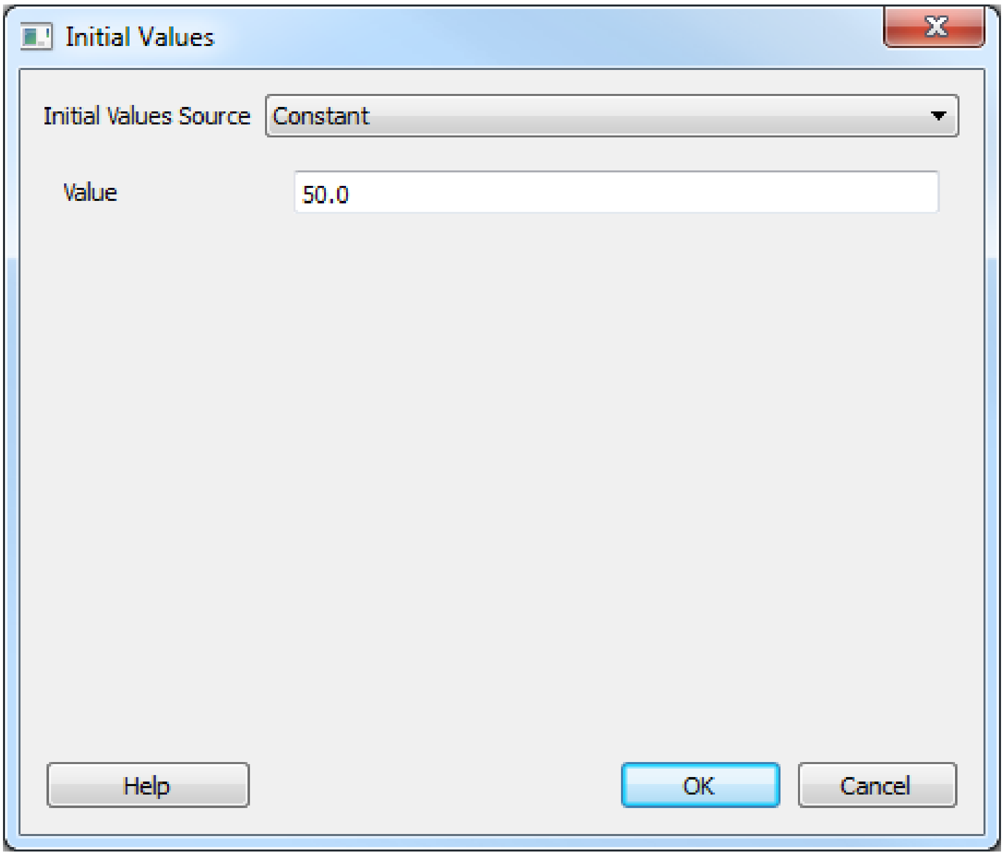
You can create a new spatial attribute by selecting the attribute name in the "New Att" combo-box. This will bring up the "Spatial Attributes Initial Values" dialog (discussed below).

The top left of the dialog has a list of the currently loaded spatial attributes. The currently selected spatial attribute can be modified by the controls on the right side of the dialog. The comment is ignored by the model and is generally used for units information. The value below the comment is intended to represent the value used the most in the dataset. The way the dataset is written to ADCIRC only the values that are different from this value need to be written. The reset button will bring up the "Spatial Attributes Initial Values" dialog (discussed below).

You can delete the current attribute by clicking on the "Delete" button. Spatial attributes cannot be deleted outside of this dialog (ie the project explorer).

Initial Values Dialog

The initial values dialog is used to setup the initial values for the dataset(s) used for the spatial attribute. Every spatial attribute has the option of starting from a constant value or use the values from an existing dataset. In addition, some spatial attributes have an option to populate the spatial attribute based upon other data inside of SMS or using a rule to generate.



Populate Options

Manning N, Chezy, and Quadratic friction coefficients

The spatial attributes which reflect bottom friction "Chezy_friction_coefficient_at_sea_floor," "Mannings_n_at_sea_floor," and "quadratic_friction_coefficient_at_sea_floor" can be populated from NLCD land use data. You can obtain NLCD geo-tiffs from the Multi-Resolution Land Characteristics Consortium website (www.mrlc.gov [1]). Use the seamless server to download the data in the area you are interested in.

When populating any of these spatial attributes, you provide a value for each land-use type and a default value that will be used in areas of the rasters that are NULL or outside the bounds of the rasters.

The values are extracted from all of the visible rasters. For each node in the mesh, the "area of influence" is computed for the node. The area of influence is a polygon that represents the area of the mesh around a node. This is computed as the area encompassing the node and half the distance to each of his neighboring nodes. All of the raster values within the area of influence are extracted from all the visible rasters. A composite n value is computed taking a weighted average of all the raster values. If the area of the extracted values is less then the area of influence around the node (because we passed the extent of the rasters or encountered null values), the default value is used to compute the composite n using the area not represented in the values.

Primitive weighting

If the average distance between a node and its neighbors is less than the critical value (default is 1750 m) then tau0 for that node is set at the tau default (usually 0.3). If the average distance between a node and its neighbors is greater than or equal to a critical value (currently set to 1750 m) then tau0 for that node is reduced to Tau Deep (default is 0.005) for depths greater than critical depth (default 10m) and Tau Shallow (default is 0.02) for depths less than the critical depth.

Surface canopy coefficient

Allows canopy coefficient values for each vegetation type. Composes the contributing vegetation types associated with each node in a mesh and assigns canopy coefficient value to the datasets for the node.

Surface directional effective roughness

Composes the contributing vegetation types associated with each of the 12 directions (30 degrees each) around each node in a mesh and assigns the 12 characteristic or composite reduction values to the datasets for the node.

Related Links

- ADCIRC
- Boundary Conditions
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Steering

References

[1] http://www.mrlc.gov/

ADCIRC/CMS-Wave Steering

NOTE: CMS-Flow (formerly known as M2D) and CMS-Wave (formerly known as WABED) are components of the Coastal Modeling System (CMS).

Related Topics

- Steering
- CMS-Flow / CMS-Wave steering

External Links

- Jun 2002 ERDC/CHL CHETN-IV-42 Coupling of Regional and Local Circulation Models ADCIRC and M2D (now know as CMS-Flow) [1]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [35]
  - Please see this forum post [2] for an explanation of ADCIRC and STWAVE steering

References

[1] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iv-42.pdf
[2] http://aquaveo.invisionzone.com/index.php?showtopic=23

LTEA

Linear Truncation Error Analysis (LTEA) was initially presented by Dr. Scott C. Hagen as part of his doctoral research at Notre Dame. Development has continued on the methodology at the University of Central Florida. The LTEA algorithm performs analysis on an existing ADCIRC mesh and its solution to help quantify the error associated with the mesh. Normally, this ADCIRC solution is taken from a "linear ADCIRC" run. This type of run is used to make the process faster and to simplify the LTEA algorithm applied to the unstructured mesh. A second phase of the LTEA process uses the error values at each node to create a relative size function covering the domain called DelX.

The LTEA Toolbox

SMS includes a graphical interface that allows the user to use the LTEA theory to guide the generation of a finite element mesh. The tool requires two inputs which must be loaded into SMS. These include:

- A bathymetry scatter set.
- An ADCIRC coverage having at least one polygon with the boundary conditions assigned.

If you wish to start the mesh generation process from an existing ADCIRC mesh, you must convert the mesh to both a scatter set and ADCIRC conceptual model, Right clicking on the mesh in the project explorer gives access to commands to perform the basic conversion. The conceptual model must be defined on the arcs created in this way.

The toolbox is accessed through the "Mesh Generation Toolbox..." from the ADCIRC menu in the Mesh Module. From the dialog that appears, select the LTEA option and click Run. The steps that follow include:

- Step 1 – Linear Run Mesh generation. In this step the user must specify the scatter set and conceptual model to be used for mesh generation. This step generates a basic mesh from the conceptual model to perform a linear run. This mesh may be saved for future reference. If a mesh already exists that is suitable for the linear run, the option to generate a linear run mesh should be turned off.

- Step 2 – Linear ADCIRC Run. In this step, the toolbox runs ADCIRC in linear mode with the M2 tidal constituent and performs an harmonic analysis on the result. If a run of ADCIRC and harmonic analysis has already been performed, the option to use existing solution data becomes available. In this case the user must specify the data sets that contain the output of the harmonic analysis. If this is not the initial pass through the mesh generation toolbox, or if the user wants to generate a "Size guideline function" from anaother source, it may be provided. If this is the case, the linear run as well as the next step "LTEA calculations" are bypassed.
- Step 3 – LTEA Analysis. The LTEA algorithm processes the harmonic analysis output and determines the relative error due to node spacing throughout the domain. It then computes a size guideline for generating a new mesh. The guideline is a dataset (a value for each node of the linear run mesh) named "DelX". The user must instruct LTEA to only perform calcualtions on the interior (no partial molecule) or to approximate the LTEA calculations right up to the boundary. (Note: extra controls exist in the dialog for tools that are under development.)
- Step 4 – Generate Final Mesh. At this point, SMS has almost all the information required for mesh generation. The user must specify the target size of desired mesh as a number of nodes and a tolerable deviation from that target. The acceptable size transitions are also specified here (see the Smooth Dataset article for a description of this).

The tool can be used repetitively to generate various meshes of the same area with varying resolution. In these cases, the first three steps should be bypassed by entering the input data in step 1 and the "DelX" function in step 3 and then proceeding to step 4.

Case Studies / Sample Problems

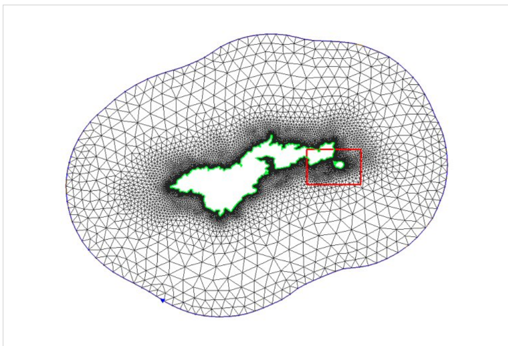
Tutorials

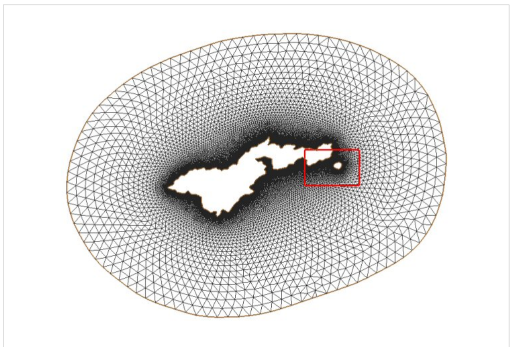
The following tutorials may be helpful for learning to use LTEA in SMS:

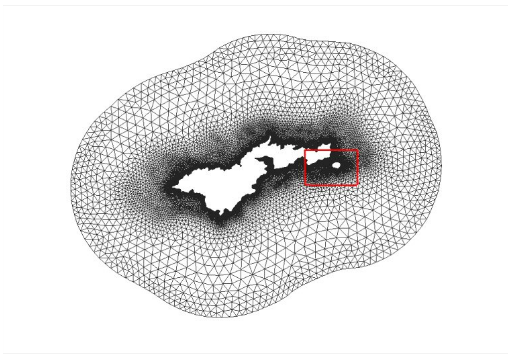
- Models Section
  - ADCIRC LTEA – Uses LTEA to mesh Shinnecock bay and the area around it along Long Island, NY

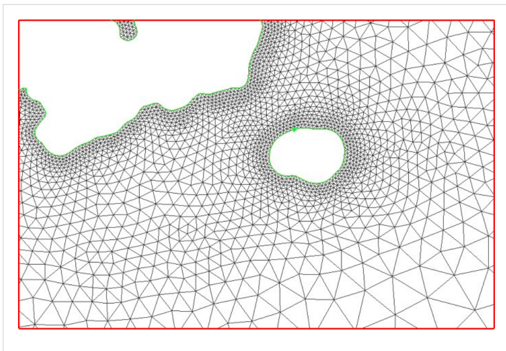
American Samoa

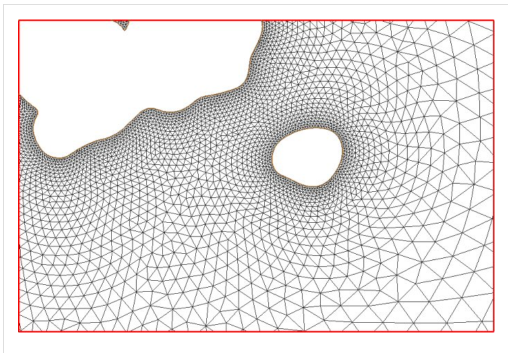
The following images illustrate the results of the LTEA toolbox applied to a domain around American Samoa. The first pair of images illustrate a mesh generated for the domain using the paving method. Density at the coastline was controlled by redistributing the vertices on the arcs representing the coastline and the density varied to a larger ocean boundary density. This mesh consists of 22,576 nodes (43,055 elements). The other images illustrate the varying resolution generated by LTEA to result in constant error with target mesh sizes of 24,000 nodes and 12,000 nodes respectively. The LTEA toolbox created meshes with 24,078 nodes (45,929 elements) and 12,029 nodes (22,543 elements).

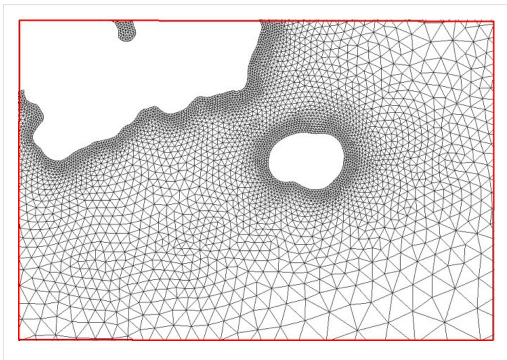








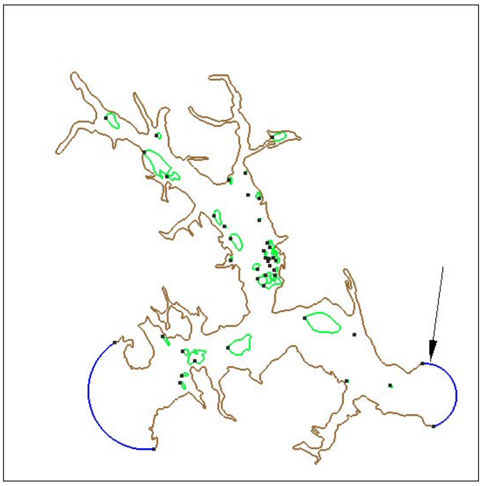




These images illustrate the redistribution of density to increase the density in areas that require additional detail for solution variations, or to reduce the number of nodes in the mesh.

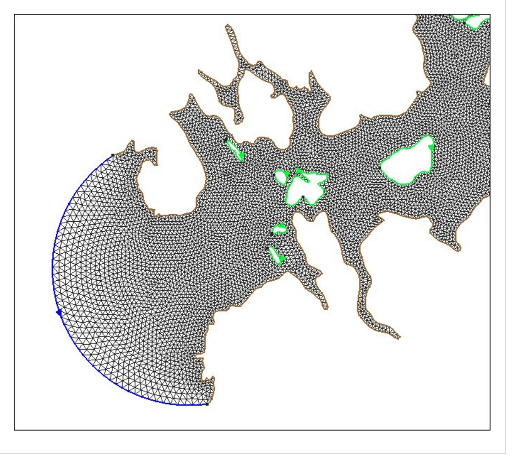
Glacier Bay Alaska

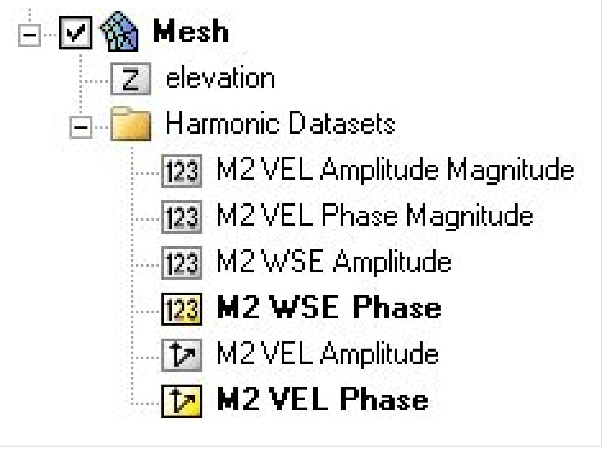
The case of Glacier Bay Alaska [7], by Dave F. Hill's [1] research group, poses another problem for the LTEA toolbox. This case includes two ocean boundaries. The figures below show three meshes generated for this case and illustrate the large variation in node density that can be produced by the procedure.



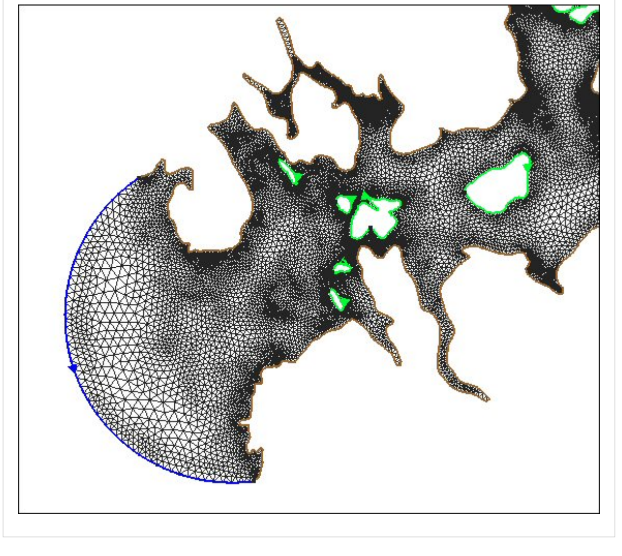
This case includes two ocean boundaries. Currently, the LTEA toolbox makes a sometimes erroneous assumption that only one ocean boundary exists. To work around this problem in the current version of SMS, the following steps are required:

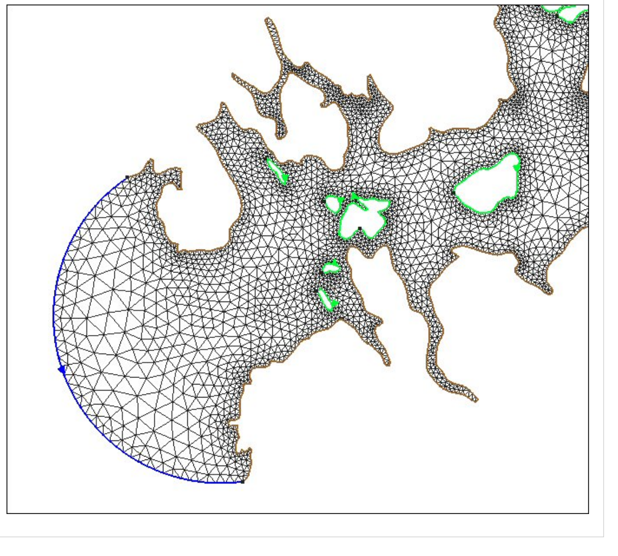
- Change the inland ocean boundary to land
- Run the first step of the LTEA toolbox to generate the "Linear Run Mesh" and then "Stop and Run" to that point.

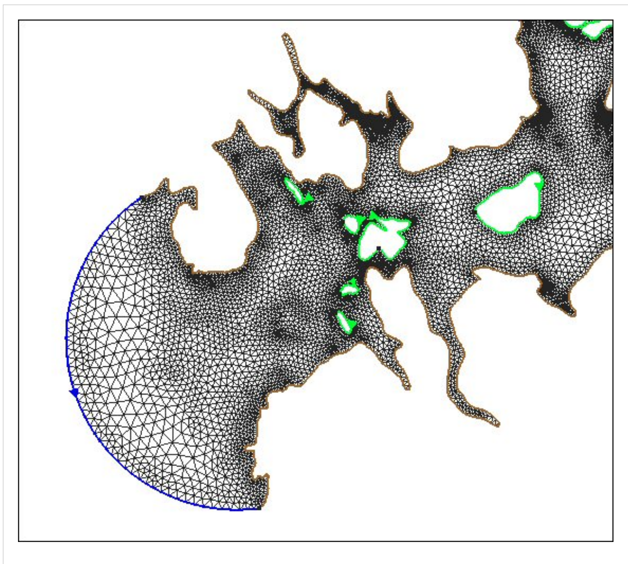




- Outside of the toolbox, change the second ocean boundary to ocean on the linear run mesh and run a linear run of ADCIRC with harmonic analysis turned on. This will generate the data sets for LTEA calcuations.
- Relaunch the toolbox and select the datasets from the linear run to guide the mesh generation process.







Related Topics

- ADCIRC
- LTEACD
- Steering

External Links

- Coastal Hydroscience Analysis, Modeling & Predictive Simulations Laboratory (CHAMPS Lab) [2]
- Dec 2006 Automatic, unstructured mesh generation for 2D, shelf-based tidal models [3]
- Sep 2006 Automatic, unstructured mesh generation for tidal calculations in a large domain [4]
- Sep 2006 Resolution Issues in Numerical Models of Oceanic and Coastal Circulation [5]
- 2001 Two-dimensional, unstructured mesh generation for tidal models [6]
- 2000 One-dimensional finite element grids based on a localized truncation error analysis [7]
- 1998 2D Finite Element Grids Based on a Localized Truncation Error Analysis [8]
- Glacier Bay Test Case by Dave F. Hill [7]

References

[1] http://water.engr.psu.edu/hill/default.stm
[2] http://champs.cecs.ucf.edu/
[3] http://contentdm.lib.byu.edu/ETD/image/etd1550.pdf
[4] http://champs.cecs.ucf.edu/Publications/Refereed/IJCFD_Hagen_et_al_2006.pdf
[5] http://poc.omp.obs-mip.fr/DOCUMENTS/t-ugo/res.pdf
[6] http://www.nd.edu/~adcirc/pubs/IJNMF-hwkh-2001-PUBL.pdf
[7] http://www.nd.edu/~adcirc/pubs/ijnmf-hwk-2000-PUBL.pdf
[8] http://kfki.baw.de/conferences/ICHE/1998-Cottbus/45.pdf

Q&A ADCIRC

Return to the Main Q&A Page

Q: Where can I find ADCIRC model documentation?

(Tags: ADCIRC, Documentation)

A: Answer.

Q: When using the XY Series Editor to create a hydrograpgh to define a nodestring boundary condition, what do the time and flowrate variables have reference to?

A: The time and flowrate variables will, most of the time, be read in from a file. They can also be user defined if no data is available. The time variable is the time when the corresponding flowrate was recorded. These times are independent from any other time variable needed by the ADCIRC model. The flowrate variable is the total flow recorded for the entire boundary specified. It is a good idea, if possible, to establish a time range in the XY Series Editor that is longer than the time to run for the sample model you are working on. This would ensure there is more than enough data for the complete model run time.

Q: How is the "Sample Interval" in the Assign Flowrates from Hydrograph menu related to the FTIMINC variable defined in the fort.20 file description in the ADCIRC user manual?

A: The "Sample Interval" variable is the FTIMINC variable defined in the fort.20 file description. The definition of this variable is the time increment (secs) between consecutive sets of normal flow boundary condition values contained in the fort.20 file.

Q: What is the "Reference WSE" value in the Assign Flowrates from Hydrograph menu?

A: The "Reference WSE" stands for Reference Water Surface Elevation. It is a specified initial water depth at the boundary to be used in calculating data such as velocity, and others.

Q: Can you define different flowrate percentages at each of the nodes in the boundary nodestring?

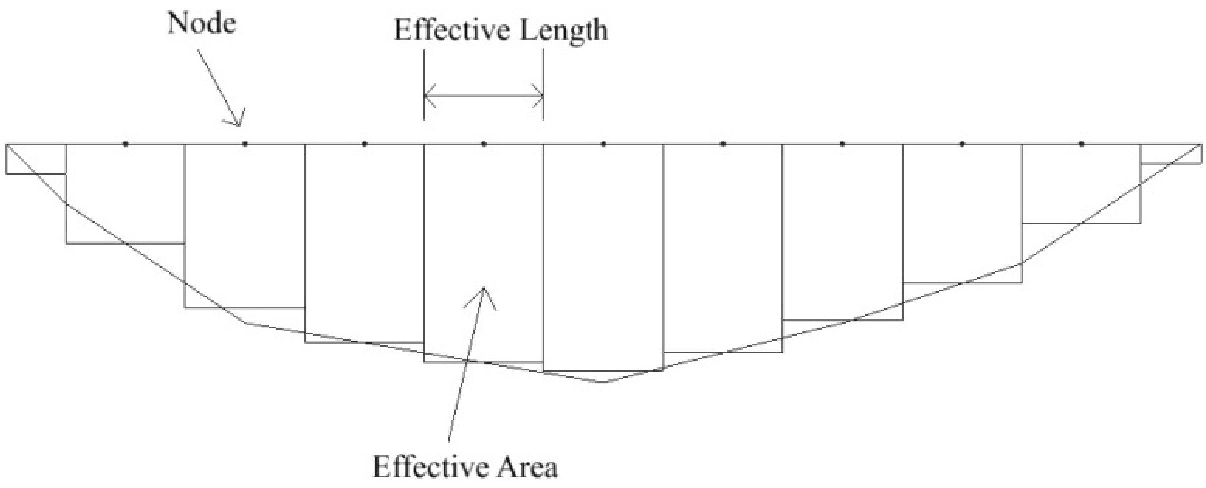
A: No, the flowrate can only be defined once for the entire nodestring. ADCIRC does, however, take into consideration the shape of the channel and the roughness, so flowrates will vary slightly across a boundary anyways.

Q: Is there a way to find the distance (or length, or width) of a nodestring?

A: Yes, there are two ways in which the length of a nodestring can be found. First, select the "Select Nodestring" tool from the toolbar. Next, select the nodestring desired by clicking on it. At the bottom of the screen in the information bar, information about the nodestring will be displayed. The length will be shown there. If you cannot see it, widen out the screen and more information should appear. The other way entails finding the distance between two nodes by selecting two nodes at once (do not select the entire nodestring, but he two end points only). The distance will appear at the very bottom of the screen in the information bar. To select two nodes at once, first select the "Select Node" tool from the toolbar, then select the first node desired by clicking on it, and last, while holding down the *SHIFT* key click on the second node.

Q: How are the normal flow rates per unit width calculated in ADCIRC?

A: Normal flow rates per unit width are calculated in the Extracted Data dialog. SMS looks at each node individually and calculates a flow rate per unit width value for each section along the boundary. It first calculates the length of each node section (effective length), the area of each node section (effective area), and the total area. Then it multiplies the total flow values defined on a hydrograph by the effective area of a node divided by the total area times the effective length of a node (or Q * (effective area/(total area * effective length))). These flow rate per unit width values should be written out to a fort.20 file when an ADCIRC model runs.



Saving ADCIRC

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.h5 referenced to new save location
- *.dat referenced to new save location

5.1.a. ADCIRC Boundary Conditions

ADCIRC Boundary Conditions

Much of ADCIRC's versatility as a model is due to the large number of different boundary types and boundary conditions available in the model.

ADCIRC Boundary Types

Boundary types are assigned to feature arcs in the conceptual mode or nodestrings on the ADCIRC mesh. Correct boundary type assignments are very important to run a successful ADCIRC project. The following boundary types are available in SMS for ADCIRC:

- **Unassigned** – default, no boundary condition assigned
- **Mainland** – land or coastline boundary
- **Island** – this can be selected for a closed feature arc
- **Normal Flow** – assigned across the nodes of the mesh, therefore the parameters of this boundary type can only be modified after a mesh has been created and the mesh module is currently selected
- **Normal Wave Radiation**
- **Mainland Barrier**
- **Island Barrier** – requires two nodestrings
- **Weir** – requires two nodestrings with the same number of nodes
- **Zero Normal Velocity Gradient**
- **Ocean**

ADCIRC Boundary Conditions

Depending upon the specified boundary type, the following boundary conditions are available:

Boundary Condition	Boundary Type
Essential w/ Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Essential w/o Tangential Slip	Mainland Island Normal Flow Mainland Barrier
Natural (w/ Tangential Slip)	Mainland Island Normal Flow Mainland Barrier
Tidal Constituents	Ocean
Curve	Ocean
Extract from Dataset	Ocean

Nodal Boundary Conditions (Recording Stations)

The following boundary conditions can be assigned to an ADCIRC mesh node:

2D Station Types

- Elevation
- Velocity
- Concentration
- Meteorological

3D Staion Types

- Density/Temperature/Salinity
- Velocity
- Turbulence

Description of Station

Stations can be assigned a name to make station identification easier.

Related Links

- ADCIRC
- Coverage
- Linear Truncation Error Analysis (LTEA)
- Meshes
- Model Control
- Spatial Attributes
- Steering

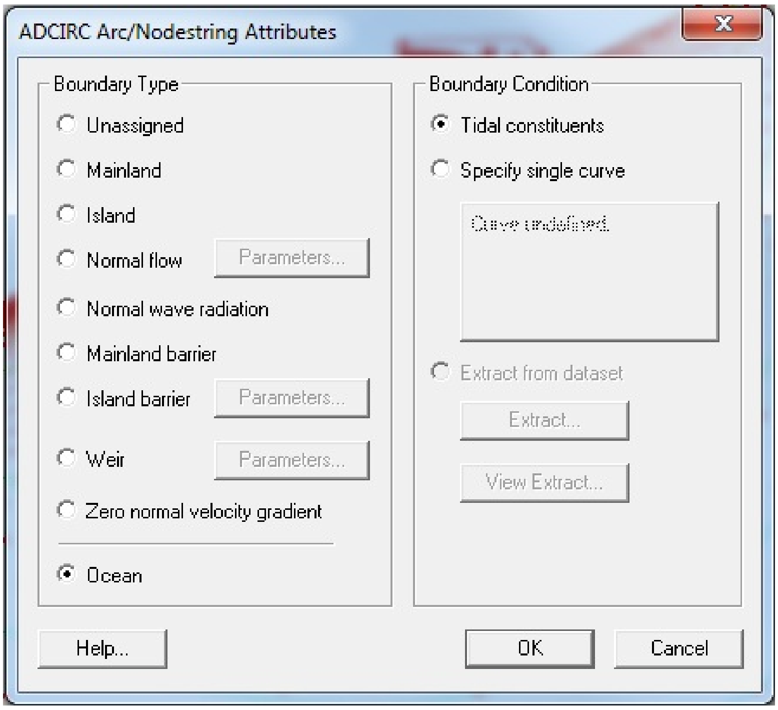
ADCIRC BC Nodestrings

Boundary conditions are assigned by setting attributes to Feature Arcs representing boundary arcs in the domain used in the simulation.

The ADCIRC Arc/Nodestring Attributes dialog allows the user to define the type of boundary conditions for each feature arc/nodestring.

A feature arc/nodestring can be defined as mainland, island, normal flow, mainland barrier, island barrier, tidal constituents, etc.

To set the boundary types, choose the Select Feature Arc tool from the Toolbox and double click the desired arc to open the ADCIRC Arc/Nodestring Attributes dialog; then assign the desired boundary conditions to the arc.



ADCIRC Arc/Nodestring Attributes dialog

ADCIRC Weirs and Island Barriers

ADCIRC Weirs and Island Barriers

An ADCIRC weir is a boundary type that are assigned to two nodestrings on an ADCIRC mesh. A weir comprises of two nodestrings next to each other with an equal amount of nodes on each nodestrings. In order to add a weir in ADCIRC, there must be two adjacent nodestrings available as shown in figures 1 and 2 below.

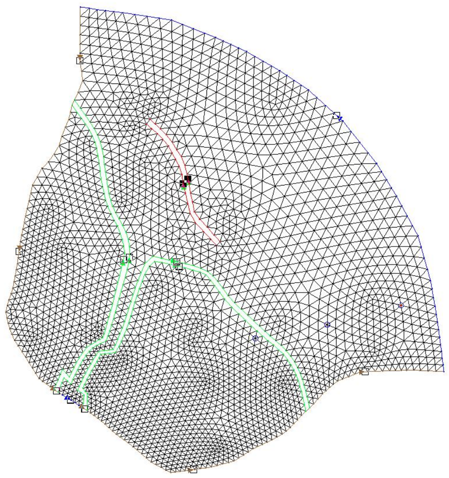


Figure 1 – Parallel nodestrings that can be made into weirs.

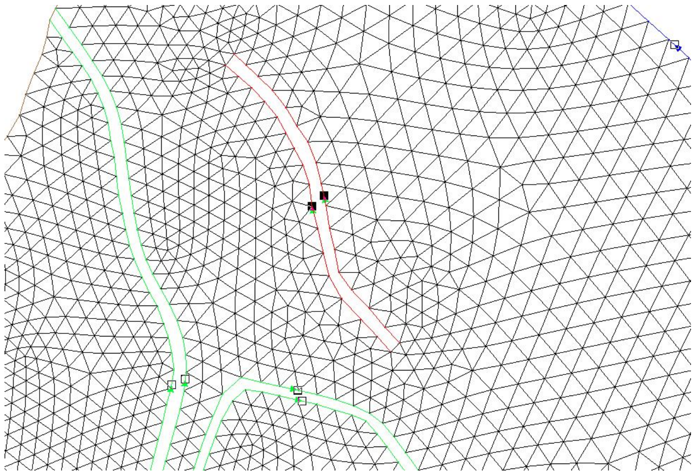


Figure 2 – Adjacent nodestrings that can be assigned as weir.

The number of nodes on each nodestrings must be of an equal amount or a weir cannot be assigned. See figure 3 below. Each node and it's corresponding node on the parallel nodestring will form a node pair.

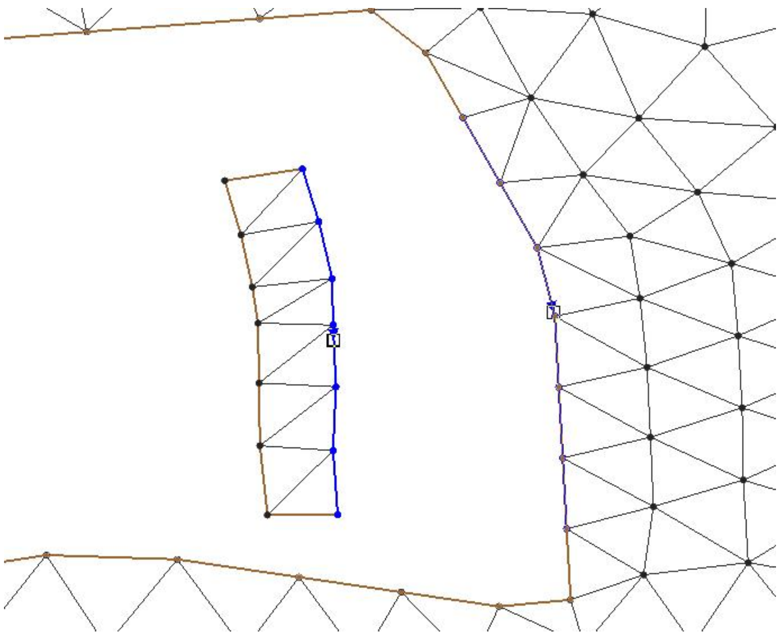


Figure 3 – Nodestrings with same number of nodes.

Weir Options

Once two nodestrings with same number of nodes exist, they can be assigned as a weir by selecting both weirs and selecting Assign BC. The Nodesting attributes will open. By selecting Weir and then clicking on Parameters button, the Weir Options dialog will open. The dialog shows the different Node pairs and elevation for each node pair.

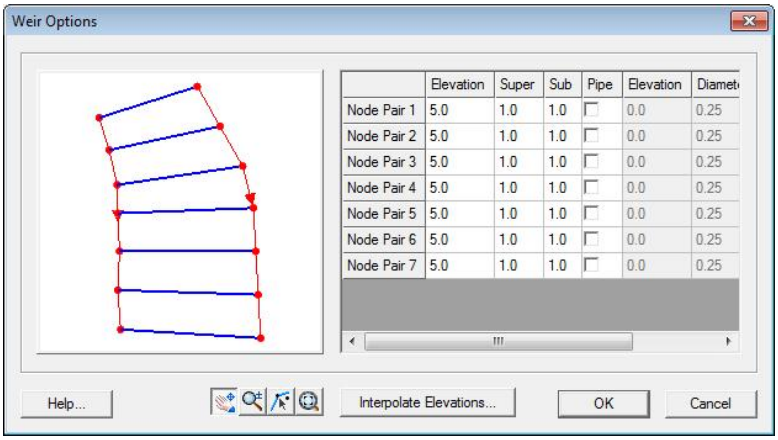


Figure 4 – Weir Options dialog.

Elevation

Elevation of each node pair. The elevation for each node pair can be entered manually or can be interpolated from a dataset

Super

Weir flow coefficients for super critical flow. A typical value for this is 1.0.

Sub

Weir flow coefficients for sub critical flow. A typical value for this is 1.0.

- Pipe
- Elevation = elevation of the pipe
- Diameter = diameter of pipe
- Coefficient = *f* (*L*/*D*)

$f \cdot \frac{L}{D}$ where

– L is the length of the pipe

– D is the hydraulic diameter of the pipe (for a pipe of circular section, this equals to the internal diameter of the pipe)

– f is a dimensionless coefficient called the Darcy friction factor [1]. It can be found from a Moody diagram [2] or by solving the Colebrook equation.

Interpolate Elevations Button

Clicking on the Interpolate Elevations button opens the XY Series Editor. The XY Series Editor can be used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use. You can also paste your xy data directly to the spreadsheet.

Extract Elevations

Data entered in as weir or island barrier boundary condition data can be used to create arcs and/or set grid cell elevations. SMS calculates the midpoint between the two nodestrings used to create a weir or island barrier boundary condition. At each of these midpoints, the boundary condition elevation is used. To use this functionality, select the Extract Weir Elevations... option from the Nodestring menu. This will open the Extract Weir Elevations dialog. Here you can choose to use only selected nodestrings or all nodestrings in the mesh. If a coverage is used, SMS will create arcs that represent the weirs on the specified coverage. If one or more grids are used, SMS will set the elevation values for all grid cells that lie underneath the weirs or island barriers by interpolation.

References

[1] http://en.wikipedia.org/wiki/Darcy_friction_factor_formulae
[2] http://en.wikipedia.org/wiki/Moody_diagram

5.2. BOUSS-2D

BOUSS-2D

BOUSS-2D	
Model Info	
Model type	Boussinesq Wave Model for Coastal Regions and Harbors.
Developer	Okey George Nawogu, Ph.D. [1] Zeki Demirbilek, Ph.D. [2]
Web site	BOUSS-2D web site [3]
Tutorials	General Section • Data Visualization • Observation Models Section • BOUSS-2D Several sets of sample problems and case studies are available. These include: • Aquaveo sample problems • Model Validataion cases from the BOUSS-2D technical report

BOUSS-2D is a comprehensive model for simulating the propagation and tranformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. It is based on Boussinesq-type equations derived by Okey Nwogu and has been under development since 1993. The equations are depth-integrated for the conservation of mass and momentum for nonlinear waves propagating in shallow and intermediate water depths.

Graphical Interface

The BOUSS-2D Graphical Interface contains tools to create and edit a BOUSS-2D simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to BOUSS-2D. If a grid has already been created for a BOUSS-2D simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to BOUSS-2D. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the BOUSS-2D Menu. See BOUSS-2D Graphical Interface for more information.

Functionality

BOUSS-2D computes nearshore wave fields including mean wave heights, mean current direction, mean water level breaking and transient representation of water levels, currents, and wave breaking.

BOUSS-2D is a comprehensive numerical model for simulating the propagation and transformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations. The governing equations are uniformly valid from deep to shallow water and can simulate most of the phenomena of interest in the nearshore zone and harbor basins including:

- Reflection/diffraction near structures
- Energy dissipation due to wave breaking and bottom friction
- Cross-spectral energy transfer due to nonlinear wave-wave interactions
- Breaking-induced longshore and rip currents
- Wave-current interaction
- Wave interaction with porous structures

The governing equations in BOUSS-2D are solved in the time domain with a finite-difference method. Input waves may be periodic (regular) or non-periodic (irregular), and both unidirectional or multi-directional sea states may be simulated. Waves propagating out of the computation domain are either absorbed in damping layers or allowed to leave the domain freely. The SI engineering units are used in BOUSS-2D calculations.

Output Options

The user can instruct BOUSS-2D to create a variety of output files. These may include spatially varied data consisting of a value for each cell in the grid, transient data defining time series at a location, or a combination of these two options (multiple time steps of data that includes a value at each cell). The output options, along with the keyword included in the parameter file to enable these options are shown below.

- Steady-state/single value spatially varied datasets
  - Significant wave height (":HS_FILE")
  - Mean currents (":MEAN_UV_FILE")
  - Mean wave direction (":THETA_FILE")
- Transient spatially varied datasets. Each output includes data from a specified start time, to a specified end time at a specified time step.
  - Water surface elevations (":SAVE_ETA_ANIMATION")
  - Transient currents (":SAVE_UV_ANIMATION")
- Time series output at specified cells (probes). BOUSS-2D saves each type of data (for multiple locations) in a single "*.ts1" file.
  - Water surface elevations (":TS_ETA_FILE")
  - Currents (":TS_U_FILE", ":TS_V_FILE") – This saves the current at a specified elevation up from the bed. Multiple elevations can be monitored.
  - Pressure (":TS_PRESSURE_FILE") – This saves the pressure at a specified elevation up from the bed. Multiple elevations can be monitored.
  - Flow rate (":TS_Q_FILE" – This saves the flow crossing a location and can be used for overtopping.

The spatially varied data may be output in to either BOUSS-2D native files or a eXtensible Data Format File (XMDF). If BOUSS-2D format is specified, the model creates "*.grd" files for each of the single value spatially varied outputs and binary data files for the transient data. The ":SOLUTION_FILE_OPTION" in the par file instructs the model to save the data in BOUSS-2D format (if set to 0), XMDF format (if set to 1) or both formats (if set to 2). When the XMDF option is specified, the ":XMDF_SOLUTION_FILE" record must also be in the parameters file along with the name of the XMDF file to store the datasets in.

Using the Model / Practical Notes

BOUSS-2D can be applied to a wide variety of coastal and ocean engineering problems, including complex wave transformation over small coastal regions (1-5 km), wave agitation and harbor resonance studies, wave breaking over submerged obstacles, breaking-induced nearshore circulation patterns, wave-current interaction near tidal inlets, infra-gravity wave generation by groups of short waves, and wave transformation around artificial islands.

As with many numerical models, BOUSS-2D can terminate or crash due to numerical instabilities. These are usually caused by problems related to the grid, the boundary conditions, or model parameters. The following lists describe common causes of instability and methods to correct them.

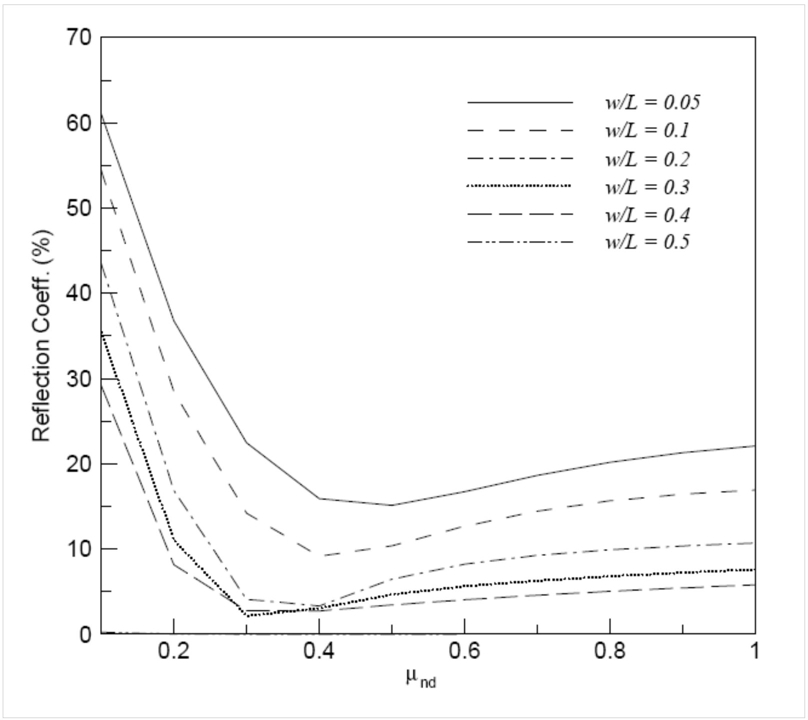
Instability due to the grid/geometry

- Model stability requires a low Courant number throughout the domain. SMS computes an approximate maximum time step to maintain a Courant number below 0.5. You may want to lower the timestep even more. Additionally, you may want to truncate the computational domain to areas with depth above a specified minimum. Another option is to increase resolution by using smaller computational cells. Either of these options increase run time, so before applying them, you may want to look at the other causes of instability.
- Abrupt changes in elevation from one cell to another in the computational domain could result in instabilities. You may find it helpful to smooth the grid. (A smoothing command is available by right clicking on the grid object in the project explorer in the SMS interface.)
- Computation nodes surrounded on three or four sides by land may be created during the grid creation process. These "isolated" cells may become unstable and generally don't have an impact on the wave climate. They can be converted to land cells.

Instability due to the boundary conditions

- Generally, avoid placing damping or porosity layers along structures and shorelines.
- Wave makers are more stable on the edges of the domain. Therefore, generally speaking, the wave maker should be placed on the boundary of the domain in constant (or nearly constant) depth water. (The SMS interface offers to extend the grid and transition to constant depth if a wave maker is created in a location with more that 20% variation in depth.) This is especially true in real world applications where reflected waves are of no concern. Also, when simulating large waves, the greater stability of external wavemakers may be required.
- Wave makers should be placed far enough from shore to avoid interaction between the wave maker and reflecting waves. This is because the external boundary behind the wave maker is treated as a vertical wall.
- Exceptions, or applications in which internal wavemakers (i.e. wavemakers placed inside the domain) are recommended include:
  - In applications with significant reflections from structures inside the computational domain. When reflected are caused by coastlines, structures, or bathymetry (reflected wave sources), the simulated seastate will become less uniform spatially, and the simulation may not reach a steady-state condition. The resulting wave field in such simulations will generally consist of nodes and anti-nodes that resemble a standing wave pattern, where waves appear to be bouncing back and forth inside the domain. If reflected waves cannot escape through boundaries of the modeling domain (or are constrained to exit the domain), a steady-state condition technically cannot be reached irrespective of the length of simulation. When reflected waves intercept external wavemakers, the extremes (lows and highs) in the calculated wavefield may keep building and can eventually lead to model instabilities.
  - Internal wavemakers should be used for finite domains and especially for limited area physical modeling studies, and with the above specified guidance.
- If wavemakers are placed on the interior of the domain, they should cross the entire domain to avoid potential "end effects", and have a damping layer placed behind (on the seaward side of) the internal wavemakers to absorb

reflected waves. There should also be a gap (at least one non-damped cell) between the internal wavemaker and the damping layer located offshore.



In the absence of laboratory or field data to calibrate damping and porous layers for an application, users should consider multiple simulations with a range of damping widths and/or coefficients. This graph from BOUSS-2D's technical report illustrates the variation of effective reflectivity given various damping coefficients and damping layer widths. To use this graph:

- Compute L (the wavelength for the incident wave).
- Select a w/L ratio. Use this ratio to compute w (damping width).
- Select an expected reflection percentage. Follow a horizontal line for this percentage on plot to intersect the graph for selected w/L ratio. Read associated damping coefficient from plot.
- Note that the reflection coefficient is very sensitive to a change in damping coefficient when the coefficient is small (< 0.3) and much less sensitive when the coefficient is larger.
- This process may require the damping parameters be changed when different wave conditions are considered.
- It should be observed that this plot is for normally incident waves. Different reflection coefficients would be obtained for obliquely incident waves.

- Damping layers should be 5-10 cells wide.

Instability due to model parameters

- The model includes a Smagorinsky term to account for subgrid turbulence. If you know turbulence is expected this term can be left at the default (0.0), however, it may be increased to increase stability. (This should be done with caution. Remember, don't suppress the wiggles, they are trying to tell you something.)

External Links

- CHL BOUSS-2D website [3]
- May 2007 ERDC/CHL CHETN-I-73 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [9]
- May 2005 ERDC/CHL CHETN-I-70 BOUSS-2D Wave Model in SMS: 2. Tutorial with Examples [25]
- Mar 2005 ERDC/CHL CHETN-I-69 BOUSS-2D Wave Model in the SMS: 1. Graphical Interface [26]
- Sep 2001 ERDC/CHL TR-1-25 BOUSS-2D: A Boussinesq Wave Model for Coastal Regions and Harbors [38]

Related Topics

- Cartesian Grid Module
- BOUSS-2D Files
- BOUSS-2D Model Control Dialog
- BOUSS-2D Calculators
- CGWAVE page
- SMS Models page
- Spectral Energy

References

[1] http://www-personal.umich.edu/~onwogu/index.html
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Persons;56
[3] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Software;23

BOUSS-2D Calculators

Run-up and Overtopping Calculator

To assist the user in the design of coastal structures, the interface includes a one-dimensional wave run-up and overtopping calculator. This utility runs a 1D simulation with BOUSS-1D based on user specified parameters. To access the 1D Run-up and Overtopping calculator select the Calculators menu item in the BOUSS2D menu to bring up the BOUSS-2D Calculators dialog. The 1D Run-up and Overtopping calculator is a tab in the BOUSS-2D Calculators dialog. The BOUSS-2D Calculators dialog is always in the BOUSS-2D interface.

Input to the 1D Run-up and Overtopping calculator is organized into a spreadsheet. The first row of the input parameters spreadsheet is fixed and will contain the column titles given in the table below. The remaining rows contain the input parameters as shown in the dialog. The names and units columns are read-only. The value column is editable.

Parameter	Value	Units
Wave Type	Choose between “Regular” and “Irregular”. Titles of the wave height & period change depending on wave type. Default is “Regular”	-
Wave Height(H)(regular) or Significant Wave Height (Hs) (irregular)	User must specify	(m)
Wave Period(T) (regular) or Peak Period (Tp) (irregular	User must specify	(sec)
Depth at Toe of Breakwater (ds)	User must specify	(m)
Crest Elevation above Still Water (Zc)	User must specify	(m)
Side Slope (m)	User must specify	-
Offshore Slope (p)	User must specify	-
Chezy Roughness Coefficient	User must specify	m1/2/sec

Output of the calculator is displayed on the bottom portion of the dialog after the calculate button is clicked. The output is organized into a spreadsheet. When the dialog first comes up, the values in the output parameters spreadsheet are blank. The output parameters calculated are:

1. Run-up, R. This value corresponds to Rmax for regular waves and R2% for irregular waves. The units of run-up are in meters
2. Overtopping, Q. A single overtopping value will be computed. The units for overtopping are (m3/s)/m.

BOUSS-2D Graphical Interface

The BOUSS-2D graphical interface includes tools to assit with creating, editing, and debugging a BOUSS-2D model. The BOUSS-2D interface exists in the Cartesian Grid Module.

Model Control

The BOUSS-2D Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In BOUSS-2D, boundary conditions are defined on cell strings. The default boundary condition is a closed boundary (no flow). See BOUSS-2D BC Cell Strings for more information.

Running the Model

The BOUSS-2D files are written automatically with the SMS project file or can be saved separately using the *File* | *Save BOUSS-2D* or *File* | *Save As* menu commands. See BOUSS-2D Files for more information on the files used for the BOUSS-2D run.

BOUSS-2D can be launched from SMS using the *BOUSS-2D* | *Run BOUSS-2D* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *BOUSS-2D* | *Model Check* menu command.

BOUSS-2D Menu

See BOUSS-2D Menu for more information.

Related Topics

- Create a 2D Cartesian grid

BOUSS-2D Menu

The following menu commands are available in the BOUSS-2D Menu:

Command	Functionality
Spectral Energy	Brings up the spectral energy dialog to define/view wave energy spectra. Generally, BOUSS-2D will generate wave conditions internally, but a spectrum may be input. This command also allows the user to visualize wave spectra that are generated inside of the model.
Assign BC	This command is used to assign damping, porosity, or wave maker conditions along a selected cell string(s).
Assign Cell Attributes	Selected cells can be defined as land or water
Probe Manager	Brings up the Probe Manager to control time series output from the model
Calculators	Brings up a pop up menu to access the Wave Conditions Calculator (see appendix A) as well as the Run-up/Overtopping Estimator.
Model Check …	Under development - check for common problems.
Model Control…	Brings up the Model Control dialog to specificy model parameters.
Run BOUSS-2D	Brings up a dialog that allows the user to check what executable of BOUSS-2D should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

BOUSS-2D Parameter File

The BOUSS-2D parameter file contains a series of comments and keywords that define the components and control the options of a simulation. Any line that begins withs "#" symbol is a comment and is ignored by the model. However, some utilities exist that may utilize comments.

Keywords begin with a ":" and are followed by values associated with the keyword.

The parameter file format is shown in Figure 1.

# ### # BOUSS-2D Run Parameter File: cirp_ideal.par # Written by: SMS # Creation Date: Tuesday June 22 16:49 2004 ###	File Header
# # Bathymetric Grid Parameters # :BATHY_FILE *filename*.grd :TIDAL_OFFSET offset value	Definition of grid and tidal offset
# # Damping Parameters :DAMPING_FILE *filename*.grd	Keywords for damping or porosity spatial input data */

# # Wavemaker #1 parameters # :START_WAVEMAKER : WM_POS_I1 value (column number) : WM_POS_J1 value (row number) : WM_POS_I2 value (column number) : WM_POS_J2 value (row number) : WAVE_TYPE value (Regular/Irreg_Uni/Irreg_Multi) : WAVE_HEIGHT value (meters) : WAVE_PERIOD value (seconds) : WAVE_DIRECTION value (degrees from North) : WAVE_CYCLES value (integer) :END_WAVEMAKER	The Wave maker definition. The position values must correspond to a single row or column. The values required depend on the wave type and method of defining that wave. The values shown illustrate a regular wave.
# # Simulation parameters # :DURATION value (seconds) :TIME_STEP value (seconds) :CHEZY_COEFF value (unitless default = 50) :SMAGORINSKY_CONST value (unitless default = 0.2) :NONLINEAR_OPTION value (Strong/Weak) :CHECK_WAVE_BREAKING value (Yes/No default - Yes) :TURB_LENGTH_SCALE value (meters0 :CALC_WAVE_RUNUP value (Yes/No default - No)	General model parameters
# # Output Parameters # :OUTPUT_FILE_PREFIX *file prefix* # # Output File for Significant Wave Height # :HS_FILE *file prefix*_hs.grd # # Output File for Mean Currents # :MEAN_UV_FILE *file prefix*_mean_uv.grd # # Output File for Mean Wave Direction # :THETA_FILE *file prefix*_theta.grd	Single Timestep (Steady State) spatially varied output options

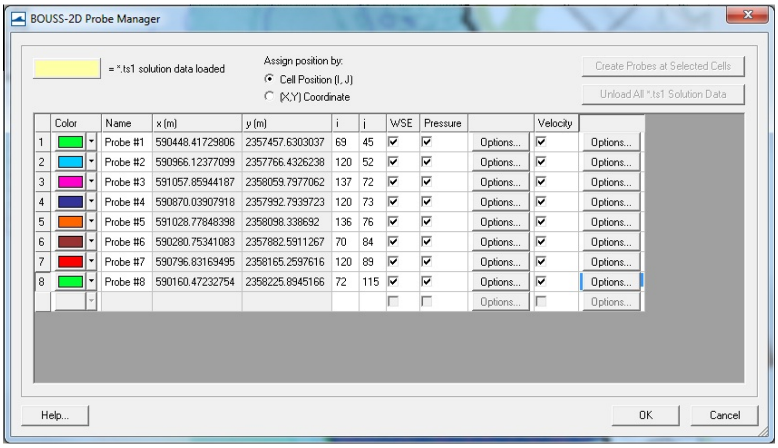
# # Surface Elevation Animation File Output Options # :SAVE_ETA_ANIMATION : ETA_ANIM_FILE *file prefix*.eta : START_TIME value (seconds) : END_TIME value (seconds) : SAVE_TIME_STEP value (seconds) : SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_ETA_ANIMATION # # Velocity Animation File Output Options # :SAVE_UV_ANIMATION : UV_ANIM_FILE *file prefix*.uv : START_TIME value (seconds) : END_TIME value (seconds) : SAVE_TIME_STEP value (seconds) : SAVE_FULL_GRID value (Yes/No default = Yes) :END_SAVE_UV_ANIMATION	Multiple Timestep (transient animations) spatially varied output options
# # Time Series Output File Names # :TS_ETA_FILE *file prefix*_ts_eta.ts1 :TS_U_FILE *file prefix*_ts_u.ts1 :TS_V_FILE *file prefix*_ts_v.ts1 :TS_PRESSURE_FILE *file prefix*_ts_pressure.ts1 :TS_Q_FILE *file prefix*_ts_q.ts1 :TS_RUNUP_FILE *file prefix*_ts_runup.ts1	Header for time series output options - must be included to create TS1 files
# # Time Series Output Options # :SAVE_TIMESERIES value (id) : SAVE_TS_ETA value (Yes/No) : SAVE_TS_UV value (Yes/No) : SAVE_TS_PRESSURE value (Yes/No) : SAVE_TS_Q value (Yes/No) : TS_X value (meters) : TS_Y value (meters) : TS_Z_UV value (meters above bed) :END_SAVE_TIMESERIES #	Time Series specification block. Repeated as many times as needed.
# # Time Series Runup Output Options # :SAVE_RUNUP value (id) : TS_X value (meters) :END_SAVE_RUNUP #	Runup Series specification block. Repeated as many times as needed. Currently only supported for 1D mode
# # Output File for XMDF Solution File # :SOLUTION_FILE_OPTION value (0 - BOUSS-2D Native, 1 - XMDF, 2 - Both) :XMDF_SOLUTION_FILE *file prefix*_sol.h5	Output format selection

Figure 1. BOUSS2D Parameter File Format.

BOUSS-2D Probes

BOUSS-2D Probe Manager

BOUSS-2D can output the histories of the computed water-surface elevation, velocities, and pressure at every grid point and at every time-step. However, due to the number of data points in the domain, this is usually done at intervals of 15-30 min.



BOUSS-2D Probe Manager dialog

In order to provide a more complete temporal representation of the results of the calculation, the model allows the user to specify probes. At a probe location (x,y,z) the user can specify what data should be saved and at what temporal resolutions. The options include water surface, velocity, and pressure.

The probe manager allows creating, editing, and deleting probes. This dialog is only accessible when a BOUSS-2D grid exists. The properties associated with probes are color, name, location, water surface elevation (WSE), pressure, and velocity.

BOUSS2D Right click Menu

If the coverage type is set to BOUSS2D | Transects, then 'Extract Elevation' appears in the right click menu of the coverage.

- **Extract Elevation –** Extracts the elevation from scattered data. To extract the elevation, arcs must first be created. The arcs must be straight lines, because bouss only supports straight line arcs.

BOUSS-2D Test Cases

Test 1 [1]

Simple test demonstrating the use of an internal wavemaker.

Test 2

Coming soon.

Test 3

Coming soon.

References

[1] http://sms.aquaveo.com/BOUSS2D-test1-internal_wavemaker.zip

Q&A BOUSS-2D

Return to the Main Q&A Page

Q: How do you select a probe on a grid?

A: To select a probe, make sure you are in the Cartesian grid module by selecting the Cartesian Grid Data menu in the project explorer. Now, select the Select Grid Cell tool from the toolbar. Then, click on the cell where the probe was defined. The probe will turn a different color to show that it has been selected.

Q: What is the meaning of the Wave Height parameter in the Spectral Parameters section of the BOUSS-2D Wave Generator Properties dialog when a Regular wave type is selected?

A: The Wave Height parameter represents the average wave height (H_{ave}). This would be the ave height of all your waves. The H_{ave} is also referred to as the RMS (root mean squared) height. The adjustment between H_{RMS} and significant wave height (H_s) is the square root of 2 or 1.414.

Q: What is significant wave height?

A: Significant wave height (or H_s) is the average height of the highest 1/3 of your waves, which works out to be about 1.6 times your average wave height (or H_{ave}).

Q: What is the meaning of the Eta Time Series plots created at a probe?

A: These plots are measurements of the surface elevation over time. They represent the free surface of your simulation.

Q: What are the _instab.grd files created with the solution of a simulation?

A: The _instab.grd files represent a data set of the unstable points of the grid where potential problems may occur. You can read these data sets into SMS and plot to better understand the meaning of them and to see where issues may arise on the grid.

Saving BOUSS-2D

When you do a *File* | *Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- Damping files saved to temp folder
- *.par referenced to new save location
- *.sol referenced to original save location unless rerun
- *.h5 referenced to new save location

5.2.a. Runup/Overtopping

BOUSS2D Runup / Overtopping

The data from various coverages in SMS is combined to create a BOUSS2D Run-up / Overtopping simulation. The different coverages are as follows:

Transects Coverage

Transects are made by creating arcs in the transects coverage. Each transect represents the 1-d grid used for a run-up simulation. It is recommended that these transects be linear. To assign elevations to the transects, use the right-click option, "Extract elevations". This will prompt you to select a dataset to use for extracting. This must be done before launching a run-up simulation.

Wavemaker Coverage

Wavemakers are made by creating arcs in this coverage. The wave maker properties can be edited by double-clicking on the arc or by right-clicking and selecting "Attributes." Each wavemaker can have multiple sets of wave parameters. Each set of wave parameters will be run in a separate simulation. It is necessary that each wavemaker arc in a simulation has the same number of wave parameters. The location of the wave makers on the transect is determined by the intersection of the transect arc and the wavemaker arc. Each transect arc may have only one wavemaker.

Probes Coverage

Probes are also made by creating arcs. The location of each probe on the transect is determined by the intersection of the transect arc and the probe arc. In the arc attributes, you can set the type of probe, which can be any of the following: Velocity, Pressure, Water Surface Elevation and Force. To create a run-up probe, simply create a polygon in the desired location. The run-up probe will be the portion of the transect arc that is inside of the polygon.

Auto-Create Probes

The properties dialog for the probes coverage can be used to automatically create probes.

Runup Probes

To automatically create a runup probe, define the following variables:

- **Minumum z –** probes will not be created below this elevation
- **Maximum z –** probes will not be created above this elevation
- **Delta z –** the difference in elevation that must occur for a probe to be created

When auto-creating these probes, SMS will traverse the transect arc, starting from the land and going seaward. Once a local maximum that is below "maximum z" is found, this point will be the starting point of the probe. SMS continues to traverse the arc until it reaches a point that is a distance of "delta z" below the starting point. This point will be the end point. However, if SMS reaches another local maximum with a higher elevation than the starting point, this point with the higher elevation will replace the original starting point.

Other Probes

To automatically create other types of probes, you need to set the following parameters:

- **Elevation (m)** – this is the elevation (along the transect) at which the probe should be placed
- **Location** – this can be set to "First" or "Last". This is used only when the defined elevation occurs more than once on the transect arc. If "First" is selected, the probe will be placed at the first qualifying location on the transect arc. If "Last" is selected, the probe will be placed at the last location.
- **Probe Types** – the available probe types are flowrate, WSE, force, pressure, and velocity. You can set which type(s) should be created for this rule. For pressure and velocity, you must define elevation(s) above the seabed where they should be placed. These can be entered by clicking the "Define..." button next to their checkboxes.

Damping / Porosity Coverage

Damping and porosity attributes are created by making arcs. The type of attribute (damping or porosity) and its coefficient and width are set in the arc properties.

Roughness Coverage

If there is varying roughness throughout a transect arc, roughness polygons can be used to define the varying roughness. Each polygon that is created will be assigned a Chezy roughness value (set in the polygon attributes). The portion of the transect arc that is inside of the polygon will be assigned that value. Otherwise, the default Chezy coefficient (defined in the model control), will be used. Note: It is helpful to turn on the display of inactive coverages while creating the wavemakers, probes, damping arcs, porosity arcs, and roughness polygons. This enables you to see the location of the transect arcs to ensure that you are intersecting the wavemakers, probes, etc., in the desired locations.

BOUSS2D Runup / Overtopping Input Files

The following files are used by Bouss2d in a run-up simulation:

- *.par
- bathy.grd
- porosity.grd (optional)
- damping.grd (optional)
- friction.grd (optional)

Each of these can be read/written by SMS. The prefix for each file will follow this format: "coveragename_arcId".

Parameter File

SMS will write a parameter file for each set of wave parameters on each transect. This file contains model control information, 1-d grid data, and the wave maker parameters. Each parameter file represents a unique simulation.

Bathymetry File

This file defines the elevations along the 1-d grid (transect arc).

Porosity File

A porosity file will be written for each transect arc that is overlapped by a porosity arc. This file defines the porosity values along the 1-d grid. If no porosity arcs are defined, the model uses a default value of 1.0 along the entire grid.

Damping File

A damping file will be written for each transect arc that is overlapped by a damping arc. This file defines the damping values along the 1-d grid. If no damping arcs are defined, the model uses a default value of 0.0 along the entire grid.

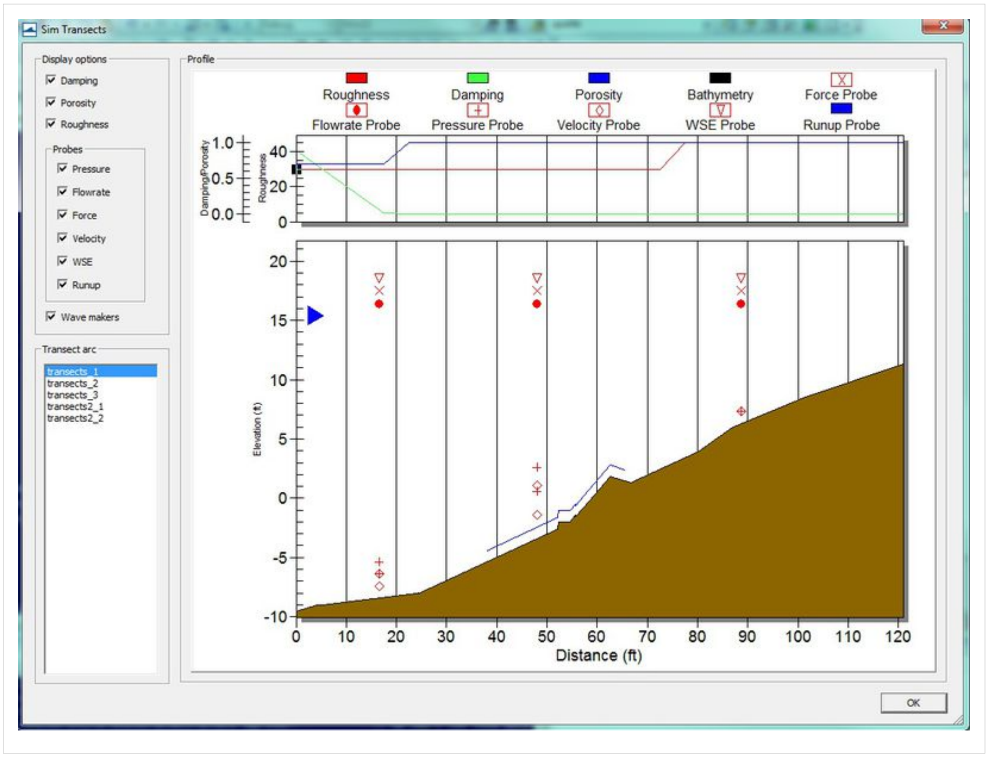
Friction File

A friction file will be written for each transect arc that is overlapped by a roughness polygon. This file defines the roughness values along the 1-d grid. If no roughness polygons are defined, the model uses the Chezy coefficient defined in the model control along the entire grid.

BOUSS2D Runup / Overtopping Viewing Data

Transect Profile

By using the plot wizard or by right-clicking on the simulation icon in the project explorer, you have the option to view the profile of an individual transect. Each arc in the simulation can be viewed by selecting the desired arc in the Transect arc list on the bottom left portion of the dialog. The bathymetry is displayed with the locations of probes and wavemakers. On a separate plot, the damping, roughness, and porosity can be viewed. This plot is for viewing purposes only.

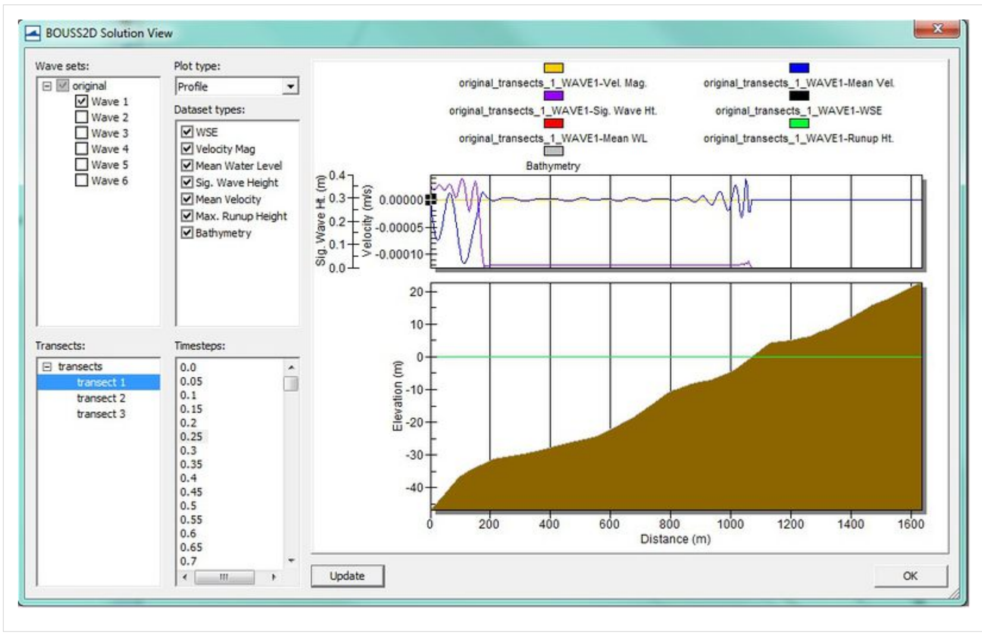


Solution View

After running a simulation, the data can be viewed in the Solution dialog. This dialog can be accessed through the Plot Wizard. There are two different views in this dialog: Profile and Time Series.

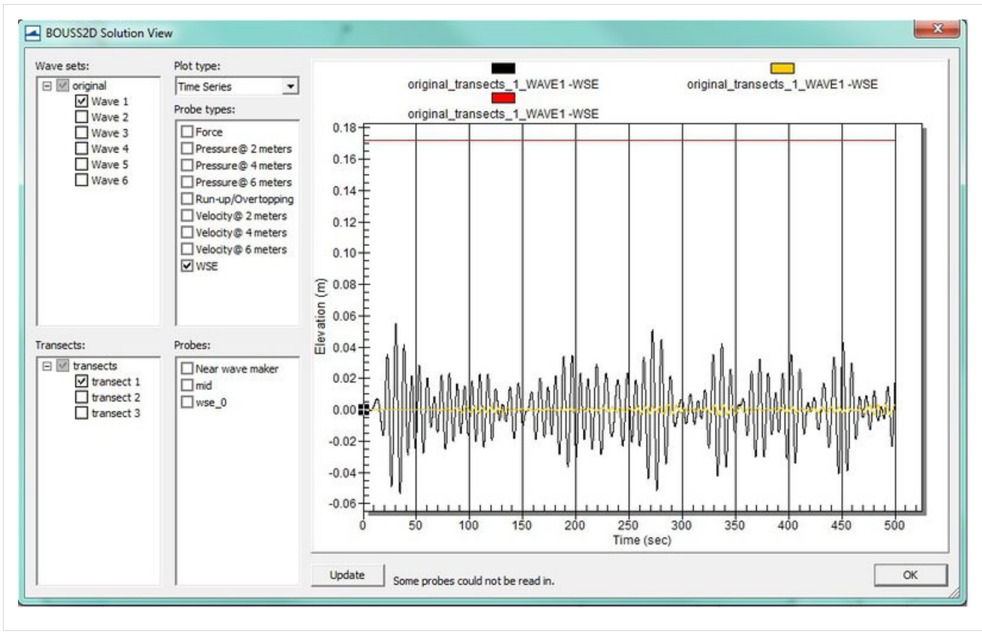
Profile

This view is used to display the bathymetry with the following datasets if they were turned on in the model control: WSE, velocity magnitude, mean water level, significant wave height, mean velocity, and maximum runup height. You can view the data for multiple wave sets for a single transect at the same time. Once the desired data is selected, clicking the "Update" button will refresh the plot window to show the selected data.



Time Series

The time series view displays the probe data from the simulation. In this view, you can display the data for any combination of transects, wave cases, and probes.



Statistics

If a simulation has been run, the "Statistics" option will be available in the right-click menu for the simulation. This dialog will display various statistics for velocity, pressure, force, and water surface elevation probes on each transect. The time series statistics display the following:

- Minimum – the minimum value in the time series
- Maximum – the maximum value in the time series
- Mean – the mean value for the time series
- Standard Deviation – the standard deviation for the time series

The "Eta Zero-Crossing" option displays:

- HAV – Average value of all peaks.
- H13 – Value exceeded by 1/3 of the peaks.
- H110 – Value exceeded by 1/10 of the peaks.
- HMAX – Maximum value of all peaks.
- TAV – Average period.
- T13 – Period exceeded by 1/3 of the peaks.
- T110 – Period exceeded by 1/10 of the peaks.

The "Runup Statistics" option displays:

- RMAX – Maximum value of all peaks.
- R2 – Value exceeded by 2% of peaks.
- R10 – Value exceeded by 10% of peaks.
- R33 – Value exceeded by 1/3 of peaks.
- RMEAN – Average of all peaks.

5.3. CGWAVE

CGWAVE

CGWAVE	
Model Info	
Model type	General-purpose wave prediction model for simulating the propagation and transformation of ocean waves in coastal regions and harbors, and appropriate for modeling the most significant physical processes in channels, inlets and harbors, open coastal regions, around islands and structures.
Developer	Vijay Panchang, Ph.D. [1] Zeki Demirbilek, Ph.D. [2]
Web site	CGWAVE web site [2]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • CGWAVE Several sets of sample problems and case studies are available. These include: • Aquaveo Sample Problems • Model Validataion cases from the BOUSS-2D website [3]

The model CGWAVE (Demirbilek and Panchang 1998) is a two-dimensional wave transformation model that can be used to predict wave properties (wave heights, velocities, pressures, radiation stresses) in domains of complex shape and depth variations when an input wave condition (amplitude, direction, and period; or a spectral combination of these) is provided.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the CGWAVE model. See CGWAVE Graphical Interface for more information.

The CGWAVE Graphical Interface contains tools to create and edit an CGWAVE simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to CGWAVE. If a mesh has already been created for a CGWAVE simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to CGWAVE. See Building a Mesh for more information.

The interface consists of the 2D Mesh Module Menus and tools augmented by the CGWAVE Menu. See CGWAVE Graphical Interface for more information.

Functionality

The model is based on extensions of the “combined refraction-diffraction” equation, which is applicable to both long and short waves and hence finds wide application in coastal engineering and harbor resonance studies. Being elliptic, the equation represents a boundary value problem, which can accommodate internal non-homogeneities (islands, structures, etc.) and boundaries. It hence forms a well-accepted basis for performing wave simulations in regions with arbitrarily-shaped (manmade or natural) boundaries and arbitrary depth variations without limitations on the angle of wave incidence or the degree and direction of wave reflection and scattering that can be modelled. In essence, it represents the complete two-dimensional wave-scattering problem for the non-homogeneous Helmholtz equation. Irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations (e.g. Chawla et al. 1998; Panchang et al. 1990; Zhao et al. 2001.)

The wave phenomena that can be simulated with CGWAVE are: bathymetric refraction, diffraction by structures (e.g. breakwaters) and the bathymetry, reflection (from structures and natural boundaries (seawalls, coastlines, etc) as well as from bed slopes), friction, breaking, and floating (fixed) docks. The model uses a triangular finite-element formulation with grid sizes varying throughout the domain based on the local wavelength; the grids can be efficiently generated using the SMS graphical interface when a bathymetry file is provided. The model allows one to specify the desired reflection properties along the coastline and other internal boundaries. It is therefore particularly well-suited for simulating waves in harbors. While the basic equation is intended for monochromatic waves, irregular (i.e. spectral) wave conditions are simulated in CGWAVE through a linear superposition of monochromatic simulations (e.g. Panchang et al. 1990; Zhao et al. 2001.) Typically, simulations involving a domain containing hundreds of thousands of finite element nodes can be performed in a few minutes on a PC.

For harbor applications, the model also uses a semi-circle (as an open boundary) to separate the model domain from the outer sea. A typical CGWAVE model domain is shown in Fig. 1. The input conditions are provided at the offshore ends of two one-dimensional cross-shore sections. (In practice, the input condition is known at the end of one of the transects. The condition at the offshore end of the other transect is obtained by appropriate phase translation.) A combination of the incident and reflected waves is computed along these transects using a one-dimensional version of the governing equation; this partial solution is then mapped on to the semicircle to force the two-dimensional model. The remainder of the solution on the boundary consists of a scattered wave that emanates from within the domain; this component is allowed to radiate out through the use of an impedance boundary condition. In the model interior, a finite-element grid is used to represent the depth field, and the coastlines (denoted by B in Fig. 1) are assigned a reflection coefficient. For open ocean applications, all variations must be included inside a circle, outside which the depths are assumed constant (Panchang et al. 2000).

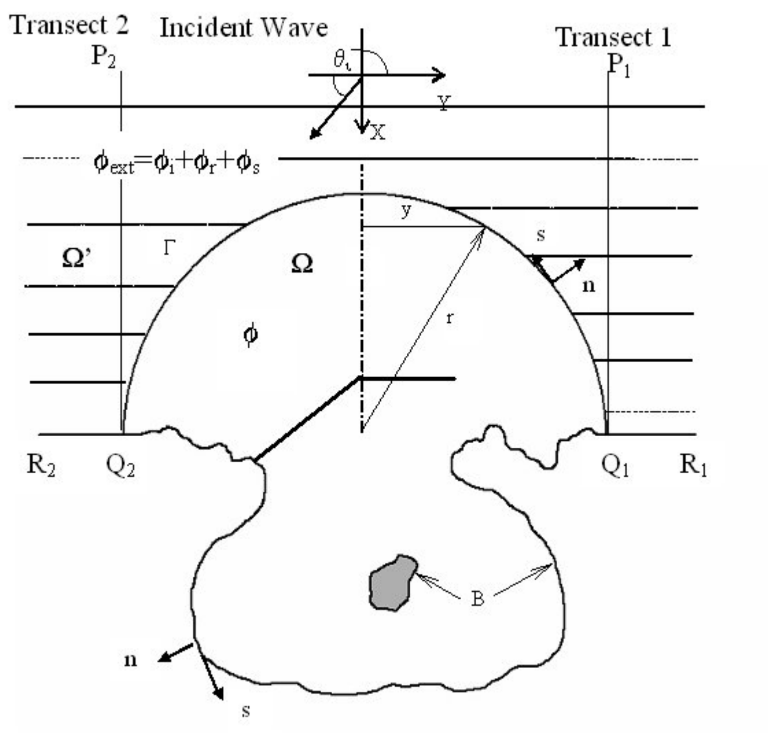


Figure 1: Harbor wave model domain; definition sketch.

Using the Model / Practical Notes

Methodology

When beginning a new project, first a domain must be created to perform the calculations. This domain comes from a combination of the coastline (land side of the domain) and the bathymetric survey. Begin by reading the survey into SMS and making sure it is an adequate and accurate representation of the study area. Ensure that the data covers the entire domain, and that the existing (or natural) coastline is adequately defined. If the survey includes topographic data, the coastline can be extracted as a contour of constant depth. Since CGWAVE does not support calculation of wave run-up, this should always be a positive depth (0.5 m or so is recommended). Follow the mesh generating techniques demonstrated in the tutorial problem for CGWAVE.

After creating the finite element mesh, it may be useful to make an initial run of CGWAVE in linear mode with fully absorbing coastlines and an incident wave condition. You may want to include both a normally incident wave run, and a separate obliquely incident wave. When CGWAVE is run in linear mode, the solution is unlimited and technically wave heights coming from the solution can grow uncontrolled. Numerically, there is no limit on wave heights. This is not realistic since waves in nature are finite in height (we know wave heights are depth-limited in shallow-water depths). However, the resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc).

Repeating the incident runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

Domain Shape

A mesh for use with CGWAVE must match one of two domain shapes. For cases involving a continuous coastline, the ocean boundary should be semi-circular. For cases examining waves around an island, the domain should be circular. The interface includes tools to build rectangular domains along coastlines, however, this is for research purposes only and should not be applied in projects with the current versions of CGWAVE. SMS includes tools in the Map Module to create a network with a valid domain shape (see the tutorial for CGWAVE). Therefore, it is strongly recommended that the Map Module be the principal method for creation of new networks. The Mesh Module may be used to create a mesh, however in the case of CGWAVE the Mesh Module primarily is useful for network editing and assigning model parameters required by CGWAVE. These model parameter commands are grouped in the CGWAVE menu.

To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axes (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. The user must specify a reflection coefficient for the exterior coastline.

Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model's exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in CGWAVE is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

Domain Size

The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using L/10 as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say $L_2/10$, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

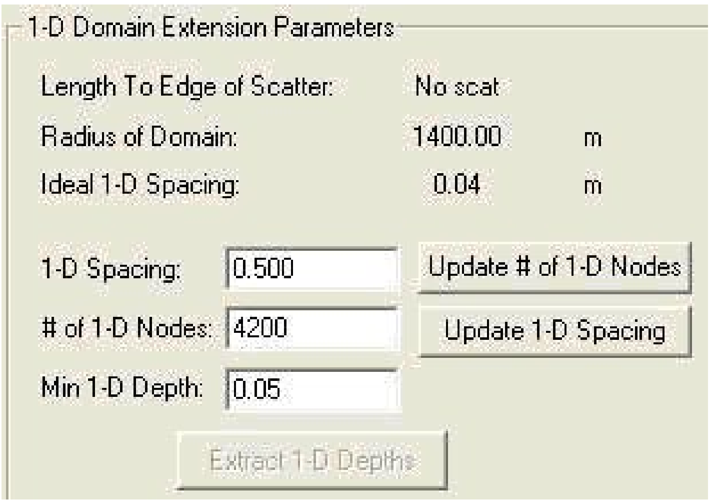
Boundary Conditions

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections MUST reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle =0°. Waves going in +y direction (i.e. to North) are specified as incident angle =90°, to West =180°, to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8}, output frequency is 100 every iterations.

For the boundary condition specification, the "parabolic plus two 1d sections" option (Option 3 in .cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in .cgi) but it really provides no benefit over Option 3. The other options consist of the "parabolic approximation" (Option 1 in .cgi) which is suitable if the exterior region is of constant depth. The

Fourier Bessel Series (Option 0 in .cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) L/10 must be specified in the dialog box:

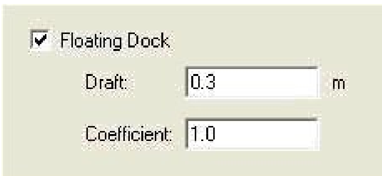


Wave Breaking

There are two methods for applying wave breaking. For practical problems, we recommend making a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say 0.78d). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6}. For spectral simulations, obviously the former approach is recommended.

Docks

For incorporating docks, the actual "without-dock" water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient as shown in the dialog below.



The draft depth is a physical quantity and can be measured on an existing structure or designed. The coefficient, which we will call alpha, is a numerical term that impacts the conveyance of energy under the dock. It is a function of the wave number of the wave passing the dock (k), the characteristic structure size (a), the depth (h) and the draft (d) of the dock. A review of the theory can be found in Tsay and Liu (1983).

To compute an appropriate coefficient, the following process can be followed:

- Compute ka, kh and d/h
- Limit value of kh
  - if (kh > 4) kh = 4
  - if (kh < 0.1) kh = 0.1
- Compute initial value of alpha as:
  - if (kh <= 2) alpha = 0.97 - (0.65*kh) + (0.1*kh*kh)
  - if (kh > 2) alpha = 0.09 - 0.02*kh

- Adjust alpha for d/h
  - alpha = alpha*(1.4-0.8*(d/h))
- Adjust alpha for ka
  - if (ka < 1) alpha = 0.8*alpha

Visualizing Solutions

Solutions generated by CGWAVE can be visualized in SMS in two ways. The most straightforward method is to read the solution file. When SMS reads this solution file, it translates the data and creates data sets for wave amplitude, direction and phase. If additional functions are desired from the wave characteristics, they must be generated using a CGWAVE translating utility. An example of this is included in the CGWAVE reference manual. The translating utility creates generic data set files that can be imported using the Data Browser.

Mesh Generation

CGWave computes properties that illustrate the characteristics of waves in a coastal area. To adequately perform this approximation, the mesh must contain several elements per wavelength. Steps for generating a mesh for CGWAVE are spelled out in the description of the Graphical Interface.

What does the error message “Number of Nodes Exceeded the Dimension” mean?

This message indicates your mesh has more elements than the program is dimensioned for. Running with more nodes/elements than the program is dimensioned for is not recommended. CGWAVE can be compiled for larger dimensions if necessary for a specific case. Contact tech support for more information.

Memory Requirements and Notes

Currently the CGWAVE engine is written in FORTRAN. All memory management in the program is handled with static arrays. An effort is currently underway to rewrite the code to use dynamic arrays. The executable distributed with SMS has been compiled to handle meshes with up to approximately 480,000 nodes. The number is not exact because some of the arrays dimensions are set based on the number of nodes on the open boundary and the coastline, which are very domain specific. In order to obtain an executable that will handle a larger number of nodes, you should contact technical support at Aquaveo. Larger executables are often posted on their ftp site.

It should be noted that due to the limitation of static arrays, the size of model that can be evaluated on a Windows PC is limited to a maximum of around 2,500,000 nodes. Users should be warned that running such a large model on a PC may take an extended period of time (possibly days).

With the availability of 64-bit machines, it is possible to access larger amounts of memory. However, this will not be an option for CGWAVE until the code is reworked to support dynamic arrays and tested on 64-bit machines.

Theoretical Basis / Mathematical Details

- Governing Equations
- Boundary Conditions
- Numerical Solution

Additional Documents:

- Simulation of Waves in Harbors Using Two-Dimensional Elliptic Equation Models [4]
- Solution of the Mild-Slope Wave Problem by Iteration [5]
- A Finite Element Model for Wave Refraction and Diffraction. Tsay, T.-K. & P.L.-F. Liu (1983). Appl. Ocean Res., v5, 1, 30-37.

- Simulation of Wave Breaking Effects in Two-Dimensional Elliptic Harbor Wave Models [6]
- Simulation of Waves at Duck (North Carolina) Using Two Numerical Models [7]
- Incorporating Rubble Mound Jetties in Elliptic Harbor Wave Models [8]
- Exterior Reflections in Elliptic Harbor Wave Models [9]
- Exterior Bathymetric Effects in Elliptic Harbor Wave Models [10]
- Improved Coastal Boundary Condition for Surface Water Waves [11]
- Outgoing Boundary Conditions for Finite-Difference Elliptic Water-Wave Models [12]

External Links:

- CHL CGWAVE website [2]
- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [9]
- Mar 2004 ERDC/CHL CHETN-I-68 How to Use CGWAVE with SMS:An Example for Tedious Creek Small Craft Harbor [31]
- Jun 2003 ERDC/CHL CHETN-I-67 Tedious Creek Small Craft Harbor:CGWAVE Model Comparisons Between Existing and Authorized Breakwater Configurations[33]
- Aug 1998 Technical Report CHL-98-xx CGWAVE: A Coastal Surface Water Wave Model of the Mild Slope Equation [43]

Related Topics

- SMS Models page

References

[1] http://www.tamug.edu/MASE/faculty_PAGES/Panchang.htm
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Software;21
[3] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=ARTICLES;447
[4] http://sms.aquaveo.com/Advances%20Coastal%20&%20Ocean%20Engg.pdf
[5] http://sms.aquaveo.com/ApplOcRes1991.pdf
[6] http://sms.aquaveo.com/Coastalengg%202001.pdf
[7] http://sms.aquaveo.com/FRF%20paper.pdf
[8] http://sms.aquaveo.com/Jetty%20paper%20final.pdf
[9] http://sms.aquaveo.com/JWPCOE-1996.pdf
[10] http://sms.aquaveo.com/JWPCOE-2000.pdf
[11] http://sms.aquaveo.com/OceanEng2000.pdf
[12] http://sms.aquaveo.com/Royal%20Society%20paper.pdf

CGWAVE Overview

CGWAVE

Overview

The model CGWAVE (Demirbilek and Panchang 1998) is a two-dimensional wave transformation model that can be used to predict wave properties (wave heights, velocities, pressures, radiation stresses) in domains of complex shape and depth variations when an input wave condition (amplitude, direction, and period; or a spectral combination of these) is provided.

Functionalities

- CGWAVE Flies
- CGWAVE Graphical Interface
- CGWAVE Model Checker
- CGWAVE Model Control
- Saving CGWAVE files

Project Explorer

Menus

The following menu commands are available in the CGWAVE Menu:

Command	Functionality
Spectral Energy	Brings up the spectral energy dialog to define/view wave energy spectra. This command also allows the user to visualize wave spectra.
Assign BC	This command is used to assign boundary conditions along a selected node string(s).
Material Properties	Material properties can be assigned and defined.
Model Check	Check for common problems.
Model Control	Brings up the Model Control dialog to specificy model parameters.
Reset 1D Spacing	...
Run CGWAVE	Brings up a dialog that allows the user to check what executable of CGWAVE should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

How do I?

To learn more about how to use the CGWAVE Model go to the Tutorials section.

Numeric Engine Background

- CGWAVE Math Details: Governing Equations
- CGWAVE Math Details: Boundary Conditions
- CGWAVE Math Details: Numerical Solution

CGWAVE BC Node Strings

In the CGWAVE model, a wave direction, amplitude and frequency must be specified at open boundaries and reflection coefficients are defined for all closed boundaries. In addition to these exterior boundaries, the model also includes the capability to simulate interior islands, and floating barriers.

Assign Boundary Condition

The CGWAVE Boundary Conditions dialog is used to assign boundary conditions to individual nodestrings. This dialog is invoked with the Assign BC Command in the CGWAVE menu. Before assigning boundary conditions to nodestrings, at least one nodestring must be selected using the Select Nodestring tool. To assign boundary conditions to the selected nodestring(s), select one of the boundary condition options.

Boundary Types

- **Open Ocean –** Delineate the region where waves will enter the domain. The attributes of the waves that enter the domain are defined as incident wave characteristics in the CGWAVE Model Control dialog. These values are propagated from an offshore location to the open ocean boundaries.
- **Coastline –** Represent a region where the wave is obstructed. At these locations, the Reflection coefficient should be set. The coastline reflection term defines to what degree a section of coastline reflects incoming waves. Legal values vary from 0.0 for a gradual sandy incline to 1.0 for solid vertical rock wall. This boundary condition may be assigned to either exterior regions of the domain, or interior holes which represent islands.

Related Topics

- CGWAVE Graphical Interface

CGWAVE Boundary Conditions

Domains on which the elliptic eq. (5) is solved are enclosed by closed boundaries (represented by coastlines and surface-penetrating structures like pier walls or pier legs, breakwaters, seawalls, etc.) and open boundaries (which represent an artificial boundary between the area being modelled and the sea region outside. A separation between the model domain and an outer water area from where no waves enter the model domain (e.g. a creek or tributary at the backbay or down wave end of the domain) may be considered to be a fully-absorbing closed boundary. An open boundary is considered to be one where an incident wave is specified (and may contain other radiated waves). Along these boundaries, appropriate conditions must be specified to solve (1); however, even in the best of circumstances, only approximate boundary conditions can be developed (e.g. see Dingemaans, 1997).

Closed Boundary Conditions

Along coastline and surface-protruding structures, the following boundary condition has traditionally been used (e.g. Berkhoff 1976; Tsay & Liu 1983; Tsay et al. 1989; Oliveira and Anastasiou 1998; Li 1994a)

$$\frac{\partial\phi}{\partial n} = \alpha\phi \quad (6)$$

where n is the outward normal to the boundary and a is related to a user-specified reflection coefficient as follows:

$$\alpha = ik\frac{1-K_r}{1+K_r} \quad (7)$$

K_r varies between 0 and 1 and specific values for different types of reflecting surfaces have been compiled by Thompson et al. (1996).

It may be verified that (6) is strictly valid only for fully-reflecting boundaries (i.e. $K_r = 1$). For partially reflecting boundaries, it is valid only if waves approach the boundary normally. For other conditions, (6) is approximate, and may produce distortions in the model solutions. These limitations may be eliminated by describing the solution at the boundary more fully as a sum of incident and reflected waves:

$$\phi = A\{\exp[ik(n\cos\theta + s\sin\theta)] + K_r \exp[ik(-n\cos\theta + s\sin\theta + \beta)]\} \quad (8)$$

where A is the amplitude of the approaching waves, θ is the direction at which they intersect the boundary (θ = 0 for normally incident waves), s is the coordinate along the tangent to the boundary, and β is a phase shift between the incident and the reflected wave. (8) leads to the following boundary condition.)

$$\frac{\partial\phi}{\partial n} = ik\cos\theta\frac{1-K_r\exp(ik\beta)}{1+K_r\exp(ik\beta)}\phi \quad (9)$$

Unfortunately, θ and β are not known a priori inside the model domain, and must be estimated by approximation. For fully absorbing boundaries (K_r = 0), Li and Anastasiou (1992) and Li et al. (1993) have used (9) after estimating θ from Snell's Law and the deep-water incident wave angle. Alternatively, Isaacson and Qu (1990) estimated θ as follows:

$$\theta = \arctan\{(\partial\chi/\partial s)/(\partial\chi/\partial n)\} \quad (10)$$

where χ is the argument of the complex quantity [Image:Theta:jpg]] (i.e. the phase of θ). For implementation, they first used (6) as a boundary condition, obtained χ from the results, determined θ from (10), used (9) as a boundary condition to perform a second iteration of the model, recalculated χ and θ , performed a third model iteration using (9), and so on. Like Pos (1985), they assumed β = 0 while using (9), based on limited numerical tests that showed little sensitivity to β . Clearly, like the Snell's Law approach, (10) is valid only for K_r = 0 (although problems with non-zero K_r were also considered). To include the effect of the reflected waves (i.e the second term in the right hand side of (8)), Isaacson et al. (1993) suggested estimating θ as follows:

$$\theta = (1/k)\arcsin\{\partial\chi/\partial s\} \quad (11)$$

Again an iterative method with repeated model calculations were needed. Steward and Panchang (2000) analyzed these methods and noted difficulties with convergence of the above iterative methods and with the quality of the solutions obtained with (10) and (11). They were able to eliminate these difficulties by estimating θ from the following expression:

$$\tan\theta = \frac{\partial\chi}{\partial s}\Bigg/\left(\frac{\partial\chi}{\partial n} + \frac{2K_r k(\cos(k\beta) + K_r)}{1 + 2K_r\cos(k\beta) + K_r^2}\cos(\theta)\right) \quad (12)$$

(12) is a generalization of (10) that allows non-zero K_r and β. For a detailed comparison of results, see Steward and Panchang (2000). Fig. 2 shows a simulation with of waves propagating into a rectangular harbor area. Clearly, solutions obtained with (12) are qualitatively superior to those obtained with (10).

Despite the increasing sophistication seen progressively in (6) and (9) and in the various ways of estimating θ, some fundamental problems remain. The most important one is inherent in (8), i.e. the assumption that the total wave field near the boundary can be represented either by one set of plane waves (in the case of (10) or the Snell's Law approach of Li et al. (1993)) or by two sets of plane waves (in the case of (11) and (12)) propagating in constant depth. In domains of complex shapes (as in Fig. 1) with arbitrary bathymetry and boundaries with varying reflectivities, a complex pattern of waves can result; simple wave trains are not easily discernible and, as noted by Isaacson and Qu (1990), the definition of a single θ (and β) can become meaningless. Further, even when there is a well-defined train of waves near the boundary (justifying the use of the above methods), precise estimation of K_r and β is still problematic. Values of K_r provided by Thompson et al. (1996) certainly do not cover full range of reflecting surfaces that the modeller encounters, nor do they cover the dependence of these parameters on the incident wave frequency. Efforts to incorporate the work of Dickson et al. (1995) and Sutherland and O’Donoghue (1998) pertaining to β in models such as the one described here are lacking.

In some ways it may be best to recognize these difficulties at the outset and use the simplest expressions (6) and (7) by combining all the uncertainties noted above into a single parameter α, which may be regarded as a tuning parameter. This is the approach followed in CGWAVE.

Open Boundary Conditions

Along the open boundary, an incident wave φ_i must be specified. Along this boundary, however, waves backscattered from within the domain will also exist, and their magnitude is generally not known. In the context of simple rectangular domain models, with one side (aligned, say, in the y direction) constituting the open boundary, Panchang et al. (1988, 1991), Li (1994a, b), and Oliveira and Anastasiou (1998) have used the following condition

$$\frac{\partial\phi}{\partial x} = ik(2\phi_i - \phi) \quad (13)$$

(13) is obtained by assuming that the incident and backscattered components along this boundary can be described by $\varphi_i = A_i\exp(ikx)$ and $\varphi = B\exp(-ikx)$ respectively (where A_i is the (specified) amplitude of the incoming wave and B is an unknown), adding the two components, and differentiating. Obviously, this is valid only if the incident and backscattered waves near the boundary are plane waves propagating in the +/- x direction.

For more complex domains involving multidirectional scattering, (13) is inappropriate. Harbor applications generally use model domains such as that described in Fig. 3, where the semicircle is used to separate the model area from the open sea. In the exterior domain Ω' the potential φ is comprised of three components:

$$\varphi = \varphi_i + \varphi_r + \varphi_s \quad (14)$$

where φ_i = the incident wave that must be specified to force the model, φ_r = a reflected wave that would exist in the absence of the harbor, and φ_s = a scattered wave that emanates as a consequence of the harbor and must satisfy the Sommerfeld radiation condition. With appropriate descriptions for these components, a boundary condition can be developed along the semicircle.

In traditional harbor models (Mei 1983; Tsay and Liu, 1983; Thompson et al. 1996; Chen and Houston, 1987; Xu and Panchang, 1993; Demirbilek and Panchang, 1998), the exterior wave conditions are described as follows:

$$\phi_i = A_i \exp\left[ikr\cos(\theta - \theta_i)\right] \quad (15)$$, which is the specified input

$$\phi_r = A_i \exp\left[ikr\cos(\theta + \theta_i)\right] \quad (16)$$

$$\phi_s = \sum_{n=0}^{\infty} H_n(kr)(A_n \cos n\theta + B_n \sin n\theta) \quad (17)$$

where (r, θ) denotes the location of a point in polar coordinates, H_n is the Hankel function of the first kind and order n, and A_n and B_n are unknown coefficients.

For the specified incident wave field given by (15), equations (16) and (17) result from the solution of the relevant eigenvalue problem in the traditional method. As demonstrated by Xu et al. (1996), however, this eigenvalue problem, in which φ_s and φ_r are coupled, may be solved only under the following conditions:

(i) The exterior region must have a constant depth,

(ii) the exterior coastlines A1D1 and A2D2 must be fully reflecting and collinear.

These requirements usually cannot be met in practice where the exterior geometry varies arbitrarily, and the unrealistic bathymetric representation used perforce by the modeller invariably has an adverse influence on the solution. In field applications, the exterior bathymetry is irregular and the depth generally increases in the x-direction. Condition (i) is thus violated, causing two problems as demonstrated by Panchang et al. (2000). First, the modeller must arbitrarily select a representative “constant” depth and test the sensitivity of the solutions to these depths. This can be extremely time-consuming. Second, the effect of reflections from the sloping exterior bathymetry is ignored. These effects are often significant, especially for long periods that are of interest in harbor resonance studies. Condition (ii) is also problematic. Exterior coastlines are not always fully reflecting for all wave conditions, and imposing full reflection in such cases yields extremely large amplification factors and rapid variations in the wave pattern in the outer regions of the domain. (See examples in Xu et al. (1996), Demirbilek et al. (1996), and in Thompson et al. 1996)). One may of course enlarge the interior region in the hope that these effects do not contaminate the results in the area of interest; however, there is no guarantee that these effects are confined to specific regions. In addition, the extra memory requirements and grid-generation for a larger domain are usually exceedingly demanding. Thus, while (16)-(17) constitute rigorous solutions of the eigenvalue problem, their use renders the application of harbor wave models problematic in practice. (One consequence of the above is that many of the models in this category cannot correctly simulate fairly simple phenomena like waves approaching a sloping beach. Investing confidence in model results when applied to field situations is therefore difficult.)

An effective alternative, followed in CGWAVE, is to use a “parabolic approximation” to describe φ_s:

$$\frac{\partial \phi_s}{\partial r} = -p\phi_s - q\frac{\partial^2 \phi_s}{\partial \theta^2} \quad (18)$$

where

$$p = -ik_0 + \frac{1}{2r} - \frac{i}{8k_0 r^2}, \quad \frac{d}{dx}\left(CC_g \frac{d\psi}{dx}\right) + kCC_g\left(k\cos^2\theta + iW\right)\psi = 0 \quad (18a)$$

where r and θ represent the polar coordinates of a point on the open boundary. (p and q are not unique, and alternative forms, each obtained with an appropriate rationale, have been investigated by Givoli (1991), Xu et al. (1996), Panchang et al. (2000)). The parabolic approximation (18) allows the scattered waves to exit only through a limited aperture around the radial direction. Unlike (17), therefore, it does not rigorously satisfy the Sommerfeld radiation condition. However, using this formulation decouples φ_s from the other components. These components (φ_i and φ_r) may be obtained by making a compromise between a detailed exterior bathymetric representation (which as noted earlier, is difficult) and the constant depth representation (which is unrealistic). A one-dimensional representation, where the depths vary in the cross-shore direction only (Fig. 1), may be selected. This is reasonable, since in general, this is often the direction in which the depths vary the most. If natural variations do not permit the representation of the exterior depths by only one section, a second one-dimensional section, shown as transect 2 in Fig. 1, may be constructed. For transects 1 and 2 with varying depths, no simple analytical expression (such as (16)) can be found for the reflected wave (since φ_i and φ_r are coupled). However, the quantity:

$$\varphi_0 = \varphi_i + \varphi_r \quad (19)$$

may be obtained by the solution of the one-dimensional version of (5), since the depths along these transects vary in one direction only. This one-dimensional equation is (Schaffer and Jonsson, 1992; Panchang et al. 2000):

$$\frac{d}{dx}(CC_g \frac{d\psi}{dx}) + kCC_g(k\cos^2\theta + iW)\psi = 0 \quad (20)$$

where, for one-dimensional geometry,

$$\varphi_0 = \psi(x)\exp(iky \sin\theta) \quad (21)$$

(20) is an elliptic ordinary differential equation requiring two boundary conditions. It may easily be solved via a simple finite-difference scheme. (For the present, the dissipation factor W is considered to be prespecified). Assuming that transect 1 extends out to a region of constant depth (or deep water), a condition at P_1 may be obtained by combining a specified incident wave

$$\varphi_i(P_1) = A_i \exp (ikx \cos\theta_i + iky \sin\theta_i) \quad (22)$$

(where A_i = a given input wave amplitude) and an unknown reflected wave:

$$\varphi_r(P_1) = B \exp (-ikx \cos\theta_i + iky \sin\theta_i) \quad (23)$$

Without loss of generality, the point P_1 may be located at x = 0, which allows elimination of B to yield

$$\frac{\partial\psi}{\partial x} = ik\cos\theta_i(2A_i - \psi) \quad (24)$$

At the coastal boundary point Q_1, the partial reflection boundary condition (9) may be used in the following form:

$$\frac{\partial\psi}{\partial x} = \frac{i\sqrt{k^2 - k^2\sin^2\theta}\,(1-K_r)}{1+K_r}\psi \quad (25)$$

where K_r is the reflection coefficient for the exterior coastline (i.e. near Q_1) and ksin#952; is constant for the one-dimensional problem.

The solution of (20) using boundary conditions (24) and (25) along with (21) produces φ_0 along transects 1 and 2. These solutions are denoted by φ_{01} and φ_{02}. The desired φ_0 along the semicircle may be obtained by laterally translating φ_{01} and φ_{02} via interpolation between transects 1 and 2 as follows:

$$\varphi_0 = (1- m)\,\varphi_{01}\exp(-ik(r-y)\sin\theta) + m\,\varphi_{02}\exp(ik(r+y)\sin\theta) \quad (26)$$

where we have set y = 0 at the center of semicircle, the interpolation function m = (r-y)/2r, r is the radius of the semicircle, y is the lateral coordinate of the open boundary node relative to the origin of semicircle (Fig.3).

The boundary condition for φ along the semicircle Γ may be obtained by using the continuity of the potential (equations 14 and 19) and its derivative along with (18) and (26):

$$\frac{\partial\psi}{\partial x} = \frac{i\sqrt{k^2 - k^2\sin^2\theta}\,(1-K_r)}{1+K_r}\psi \quad (27)$$

Thus, the solution of (20) provides φ_0 along the 1-dimensional transects. These values can be translated laterally and substituted into (27) to obtain the open boundary condition for the two-dimensional equation (1). Zhao et al. (2000) and Panchang et al. (2000) have demonstrated that this procedure provides extremely satisfactory solutions for a large number of test cases.

In CGWAVE, if non-linear breaking is implemented, a version of (21) is solved by iteration since W depends on the (unknown) amplitude.

Related Topics

- CGWAVE
- Governing Equations
- Numerical Solution

CGWAVE Boundary Conditions Dialog

The CGWAVE Boundary Conditions dialog is used to set the attributes for feature arcs. Attributes that can be specified for each arc include:

- Boundary Type
  - None
  - Open ocean
  - Coastline
    - Reflection coefficient
  - Floating Barrier

Related Topics

- CGWAVE
- CGWAVE Boundary Conditions
- Feature Objects Menu

CGWAVE Files

Input Files

The *.cgi File

CGWVE always requires an input file to run. This input file is created by SMS with an extension of *.cgi (CGwave Input). It is an ASCII file that defines the model parameters, the boundary definition, the nodal locations and the connectivity. Lines that begin with a "%" indicate the line is a comment. These are terminated with a "&" character. There is a comment for each value on the data line, so five comment lines indicate the following data line will have five values. This format is generally followed throughout. Some utilities available for processing CGWAVE data files require that all the "standard" comments exist. The model parameter data is sequentially read, and follows the following format:

%number of characters in title &

```
%number of terms in the fourier-bessel series solution (default = 35) &
%output echo frequency to screen &
%maximum number of iterations for convergence &
%max no of iters. for nonlinear mecha (fri,breaking,dispersion) &
         30          35         100      500000          10
```

Title of this Run, jul 10 2007

```
%type of application &
%type of open boundary condition &
%bottom friction &
%wave breaking &
%nonlinear dispersion relation &
%choice of solver &
          0           3           1           1           0           2
%number of components &
          1
%Incident Wave Angle || Wave Period(s) || Incident Wave Amplitude &
 225.0000000000000E+00   30.00000000000000       1.000000000000000
%exterior reflection &
%bottom friction coefficient &
%wave breaking parameter &
%tolerance for the equations &
%tolerance for nonlinear mechanisms &
  0.00E+00    0.12E+00    0.15E+00    0.10E-09    0.10E-06
```

After the model parameter data comes the boundary definition. This includes two blocks of data marked with "&C" to indicate a boundary open to have waves enter, and "&B" for sections of coastline and islands (land boundaries). There can only be one "Open" boundary (&C). Its type will be 0 or 1 indicating semi-circle or full circle. The full circle is applied when modeling waves around an island. Multiple closed boundaries (&B) may be included. One for each segment of the coastline (changing reflection coefficients), and at least one for each island in the domain.

%Open boundary flag

```
%type of open boundary &
%number of nodes on it &
%center of the circle - x &
%center of the circle - y &
%initial angle &
%semicircle orientation angle (ccw from +x axis, E=0, N=90, W=180, S=270) &
%node id, ... ...
```

&C

```
0  220 0.000000000E+00 0.000000000E+00 0.000000000E+00 0.230036700E+02
         220          219          218          217          216          215
         214          213          212          211          210          209
         208          207          206          205          204          203
         .
         .
         .
         .
         .
```

```
%Coastline boundary flag
 %number of nodes on it &
 %reflection coefficient &
 %node id, ... ...
```

&B

```
          360  5.0000000000000003E-02
            1          221          440          659          880          881
         1103         1329         1559         1793         1794         2036
         2285         2538         2798         2799         2800         3070
           .
           .
           .
           .
           .
```

Following the boundary definition, the file includes the locations of the computational nodes with a "&N" identifier:

```
%Node coordinates and Depth &
 %number of nodes &
```

&N

```
     236211
-0.452985572E+04  0.259162993E+04  0.215026303E+00 -0.452888341E+04  0.259197240E+04  0.242541197E+00
-0.452784808E+04  0.259233707E+04  0.271839587E+00 -0.452671893E+04  0.259273479E+04  0.303792731E+00
-0.452546078E+04  0.259317794E+04  0.334700922E+00 -0.452409490E+04  0.259365905E+04  0.361240031E+00
        .
        .
        .
        .
        .
```

Following the nodal locations comes the material definitions with a "&M" identifier. Material types define two important options for CGWAVE. First, roughness is specified as a material. Second, floating breakwaters are defined in the material block. Each type of breakwater, and each roughness zone will require it's own material type. The materials are then assigned to each element in the element block below. In the example below, two materials are defined. The first has a roughness of 0.5 and is type 0, indicating it is not a floating breakwater. The second has the same roughness and is of type 1, indicating it is a floating breakwater. For the floating breakwater, two additional parameters are included, a draft and a coefficient of transmission.

```
%Materials
%number of materials &
%material id &
%bottom friction &
%floating dock &
%draft &
%coefficient.
```

&M 2 1 0.5000 0 0.0000 0.0000 2 0.5000 1 0.0000 1.0000

Following the material definitions comes the element connectivity with a "&E" identifier. Each line includes the three node identifiers that make up an element and the material type that fills that element.

```
%Elements
 %number of elements &
```

&E

```
      456220
           1         221           2        1
         221         222           2        1
         222           3           2        1
        .
         .
         .
         .
         .
```

The *.cg1 File

When running CGWAVE from for varied sloped, semi-circular boundaries, it is recommended that the two "Semi 1D Lines" option be used. This option transforms the input wave conditions from the offshore depth to a variety of depths as those conditions propagate shoreward. The transformed conditions are mapped to the variable depth boundary of the semi-circle (open boundary). SMS extracts depths from the surrounding survey if it exists. Otherwise the user must create a profile from the deep water location where the wave conditions are sampled, through to the coastline. The file format includes two such profiles. One marches down the left side of the semi-circle, and the other down the right. The format of each profile includes the distance between sample points and the number of sample points. Two other values are included as space holders for now. SMS always writes out "0.0 1" for these values. The file then includes one line for each sample point including a depth and an optional roughness value at that depth as follows:

```
0.5000 4912  0.0  1
   73.394101    0.120000
   73.223910    0.120000
   73.053719    0.120000
         .
         .
         .
         .
   9.177065    0.120000
   7.433389    0.120000
   5.914170    0.120000
```

Output Files

CGWAVE creates a single ASCII solution file. It includes a block of data for each input wave condition. The file includes a complex number that defines the wave amplitude and phase at each node for each wave condition. Upon trying to read this file into SMS, the CGWAVE - Trans post-processor will be executed in order to turn the .cgo solution file into an XMDF file that can be easily read by SMS.

CGWAVE Governing Equations

Governing Equations

In its basic form, the methodology is based on solving the following two-dimensional elliptic equation:

$$\nabla \bullet \left(CC_g \nabla \phi\right) + k^2 CC_g \phi = 0 \quad (1)$$

where

$\phi(x,y)$ = complex surface elevation function $(= \phi_1 + i\phi_2)$

i = $\sqrt{-1}$

σ = wave frequency under consideration

C(x,y) = phase velocity = σ /k

C_g(x,y) = group velocity = $\partial\sigma / \partial k$

k(x,y) = wavenumber (= 2 π /L), related to the local depth d(x,y) through the dispersion relation:

σ^2 = gk tanh (kd) (2)

The wave height H can be obtained from complex surface elevation function ϕ as follows:

H = $\partial\sigma / \partial k$ (3)

Essentially (1) represents an integration over the water column of the three-dimensional Laplace equation used in potential wave theory. The integration, originally described by Berkhoff (1976) and Smith and Sprinks (1975), is necessary because the solution of the three-dimensional problem is computationally difficult for harbors with a characteristic length that is several times the wavelength. The integration is based on the assumption that the vertical variation of the wave potential is largely the same as that for a horizontal bottom, i.e.

$$\varphi(x,y,z) \approx \frac{\cosh k(d+z)}{\cosh kd}\phi(x,y) \quad (4)$$

This approximation is obviously valid for a "mild slope", characterized by $|\nabla d|$ /kd << 1, a criterion that is usually met in practice. (Extensions to steep slopes are described later). Unlike "approximate" mild slope wave models (e.g. REFDIF and RCPWAVE described by Dalrymple et al. 1984; Kirby, 1986; and Ebersole, 1985), there are no intrinsic limitations on the shape of the domain, the angle of wave incidence, or the degree and direction of wave reflection and scattering that can be modeled with (1). While (1) is valid for a monochromatic (single incident frequency-direction) wave condition, irregular wave conditions may be simulated using (1) by superposition of monochromatic simulations.

As noted earlier, (1) incorporates the effects of refraction, diffraction, and reflection induced by any nonhomogeneity in the model domain. We now provide extensions of (1) that include, in addition, dissipative effects (friction and wave breaking), steep-slope effects, and floating docks.

Dissipation

To include dissipative effects, we consider the following extended form of (1):

$$\nabla \bullet \left(CC_g \nabla \phi\right) + \left(k^2 CC_g + iC_g \sigma W\right)\phi = 0 \quad (5)$$

in which a dissipation term (with W) has been included. By separating the real and imaginary parts of (5), Booij (1981) has shown that (5) satisfies the energy balance equation in the presence of dissipation. The term W may represent breaking and/or friction and is described later.

In (5), W represents the combined effects of friction and breaking, which may be separated as follows:

$$W = w / C_g + \gamma \quad (6)$$

where w is the friction coefficient defined by Dalrymple et al. (1984) and g is a breaking factor. These coefficients are empirical, and parameterizations for these have been described by Dalrymple et al. (1984), Tsay et al. (1989), and Chen (1986) for friction and by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) for breaking. Some of these parameterizations have been extensively validated against field data (e.g. Larson 1995; Kamphuis 1994). The parameterization used in CGWAVE is based on the formulation by Dalrymple et al. (1984).

Published studies demonstrating the effects of friction in harbor models (e.g. Chen 1986; Tsay et al. 1989; Demirbilek and Panchang 1998; Kostense et al. 1986) have estimated w on the basis of the incident wave amplitude. It is then easy to pre-specify w while solving (5). These studies appear to show that friction can change the magnitude of resonant peaks in harbor models quite substantially; at other frequencies, the effect seems to be minimal.

As to breaking, Zhao et al. (2000) applied a finite element wave model to several tests involving breaking. These tests involved a sloping beach, a bar-trough bottom configuration, shore-connected and shore-parallel breakwaters on a sloping beach, and two field cases in the North Sea and Ponce de Leon Inlet (Florida). Five breaking formulations, given by Battjes and Janssen (1978), Dally et al. (1985), Massel (1992), Chawla et al. (1998), and Isobe (1999) were examined. In general, they found that the formulations of Battjes and Janssen (1978) and Dally et al. (1985) were the most robust from point of view of incorporation into an elliptic model based on (5) and provided excellent results compared to data. CGWAVE includes the Battjes and Janssen (1978) formulation.

Unlike friction the breaking coefficient is a function of the wave amplitude which is unknown *a priori* inside the domain, and its inclusion makes the problem nonlinear and requires iteration. For the first iteration, W is set equal to 0 and (1) is solved (e.g. non-breaking solutions are obtained). The resulting wave heights are used to estimate W via the Battjes and Janssen (1978) parameterizations and (5) is solved again. The process is repeated until convergence is obtained. This can be a very time –consuming process. For many practical applications, it is therefore suggested that simple non-breaking simulations be performed, and an H/D breaking limit be then applied to artificially cut off the excessively large waves. This option is available in CGWAVE.

Steep-slope effects

Unlike the nonlinear mechanisms described above, the “mild slope” requirement discussed in Section 1 is relatively easy to eliminate. Massel (1993), Porter and Staziker (1995), Chamberlain and Porter (1995), and Chandrasekera and Cheung (1997) developed extensions of (1) to include steep-slope effects. Their extensions may be described by the following equation:

$$\nabla \bullet \left(CC_g \nabla \phi\right) + \left(k^2 CC_g + d_1 (\nabla h)^2 + d_2 \nabla^2 h\right)\phi = 0 \quad (35)$$

where d_1 and d_2 are functions of local depths. Reference may be made to these publications for the various definitions of d_1 and d_2; in general, though, differences in the proposed definitions of these functions impact model results to a very small extent. The steep-slope terms are fairly straightforward to include in the model because they are linear. Further, they have the advantage of being “automatic”, i.e. they have little contribution for mild slopes and do not change the solution technique; when they are significant, the additional computational demand is negligible. However, steep slopes lead to breaking and model performance in the vicinity of steep slopes (e.g. see the work of Massel and Gourlay (2000) that include breaking and steep-slope effects near coral reefs). CGWAVE uses the formulation by Chandrasekera and Cheung (1997).

Floating Docks

One problem frequently encountered by engineers when using models based on (1) pertains to the presence of floating structures in the modeling domain (e.g. floating breakwaters or docks in marinas). These structures of course violate the “free-surface” requirement of (1). The problem near the dock is 3-dimensional, whereas the model (eq. 1) is solved in a 2-dimensional framework. CGWAVE uses an approximate method suggested by Tsay & Liu (1983) for tackling floating structures in the context of 2-d harbor wave models. This approach merely calls for a suitable modification to the second term on the left-hand side of (1). (Tsay and Liu (1983) examined suppressing this term). As a consequence, the method is extremely simple to implement with existing finite element models. A model grid is first generated as usual with no regard to the floating structure, grid elements covering the floating structure (in plan view) are selected, they are assigned a depth value equal to the under-keel clearance, and the coefficient of the second term in (1) is set to zero for these elements. Clearly, this is an ad-hoc method intended for convenience in engineering practice, and although Tsay & Liu (1983) provided heuristic arguments in support of this approach, their testing of this procedure was rather limited. Li et al. (2005) found, however, that the method produces results which deviate considerably from the solution of the Laplace equation, and hence developed a simple modification to the original Tsay & Liu (1983) approximation. This involves adjusting the under-keel depth by a factor $\alpha = A\ln(ka)+B$, where a = half width of the structure, and A and B are given in Figure 2 (and Table 1) for different values of relative submergence (defined by draft/water-depth = d/h). The modified approximation yields improved results, when compared to both laboratory data and theoretical results, for a wide range of conditions. By way of practical demonstration, simulations in Douglas Harbor (Alaska) was described by Li et al. (2005) for examining the effects of proposed floating dock configurations. The factor α must be provided to CGWAVE along with the draft depths during grid generation.

Related Topics

- CGWAVE
- Boundary Conditions
- Numerical Solution

CGWAVE Graphical Interface

The CGWAVE Graphical Interface includes tools to assist with creating, editing, and debugging a CGWAVE model. The CGWAVE interface exists in the 2D Mesh Module.

Model Construction Steps

There is a very consistent method that can be used to apply the CGWAVE model. The steps to this process include:

1. Load bathymetry – This data can come from LIDAR surveys, digital elevation maps (DEMs), previous grids or a variety of other sources. They must be referenced to the same datum the wave data will reference, and must have positive values represent depths. SMS includes functionality to convert datums, reverse directions and smoothor filter data.
2. Limit bathymetry to positive values – CGWAVE does not handle wetting/drying or runup processes. All the nodes in the model must have a positive depth. This limiting process can be handled later in the mesh generation process, or the bathymetry itself can be modified (in a copied dataset) using the data calculator. For example, if you want to limit the domain to areas of at least one meter of depth use the following equation in the data calculator *max(d1, 1.0)* (where d1 is the label for the depth dataset).
3. Compute the wave length – This is also done in the data set toolbox, using the Wave Length and Celerity tool. Enter the smallest wave period of interest (to generate the highest needed resolution).
4. Create a size function – Typically this is simply a scaled version of the wave length data set. For example, a basic rule would be to have at least six elements per wave length. Some experts recommend at least 10 elements per wave length. The real issue is that you need enough resolution to represent the wave shape.
   1. To create a "Size" function with "N" elements per wave length, go to the data calculator and enter the equation *d3/N* (where d3 is the label for the wave length dataset).
   2. An alternative that may be needed with large domains is to create a spatially varied scale for the wave length function. For example, you may want to have 15 elements per wave length in the shallow region or your model (i.e. depths less than 3 meters), but only have 7 elements per wave length in the deep regions (i.e. depths greater than 200 meters). To create this size function, you would still use the data calculator and enter the equation *d3/max(min(15+(d1-3)/(200-3)*(7-15),7))* (where d1 is the label for the depth function and d3 is the label for the wave length dataset).
5. Define the coastline or land edge of the domain – This can be done using a contour of the bathymetry or reading a coastline vector file. It should be stored in SMS as a coastline arc in a CGWAVE coverage in the map module.
6. Define the ocean boundary – This is also an arc in the CGWAVE coverage. It must be either a semi circle or circle and can be defined in SMS by selecting the coastline (or extreme locations on the coastline) and issuing the *Feature Objects | Define domain...* command in the map module.
7. Build polygons in the map module and assign polygon attributes for the polygon to use the depth and size functions for bathymetry and size respectively in the mesh generation process. Then generate the mesh.
8. With this mesh constructed, the rest of the graphical interface, defined below can be used to control the numerical simulation.

Model Control

The CGWAVE Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. In CGWAVE boundary conditions are defined on nodestrings. The default boundary condition is a closed boundary (no flow). See CGWAVE BC Nodestrings for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Running the Model

The CGWAVE Files are written automatically with the SMS project file or can be saved separately using the *File | Save CGWAVE* or *File | Save As* menu commands.

CGWAVE can be launched from SMS using the *CGWAVE | Run CGWAVE* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *CGWAVE | Model Check* menu command.

CGWAVE Menu

See CGWAVE Menu for more information.

Processing Solutions

CGWave creates a single output file (normally including ".cgo" extension when run with SMS). This file can be brought into SMS to graphically view the results. As SMS reads the file, it translates (using an embeded version of the CGWAVE "trans" code) the complex numbers representing wave heights and phases for each wave component into spatial and temporal data sets including:

- Steady State
  - Wave Height
  - Wave Phase
  - Direction of Maximum Particle Velocity
- Time Varying (through a single wave period. SMS breaks the period into 20 timesteps)
  - Pressures (at surface, mid depth and bed)
  - Particle Velocities (at surface, mid depth and bed)
  - Sea Surface Elevation
  - Wave Velocity.

When the solution is read in, SMS allows the user to limit the wave heights in the solution. This only applies to linear runs of CGWAVE to allow the heights to be adjusted to be more realistic. This is accomplished by applying a factor, whose value ranges from 0 to 1 (defined as H/d). (Because wave height cannot exceed the value of depth or factor=1). SMS recommends a range of 0.4 to 0.8 and defaults to 0.64. The user may enter another factor dependent on that users knowledge of wave mechanics, type of problem, etc.

Related Topics

- CGWAVE
- SMS Models

CGWAVE Incident Wave Conditions

The Incident Wave Conditions section allows the user to specify wave conditions to be simulated in a run. Each condition is defined as a wave direction, period and amplitude. The wave direction is specified in degrees measured from the positive x-direction in a counter clockwise direction. This means that waves propagating from West to East have a direction of 0°, and waves propagating from East to North have a direction of 90°. The amplitude is specified in meters, and the period in seconds. These values are used as boundary conditions out to sea and are propagated to the open ocean boundaries on the domain using one-dimensional wave propagation. A set of values (one direction, frequency and period) can be selected by clicking in the window. The selected set can be via the edit boxes above the window. The buttons below the window can be used as follows:

- New – The New button creates a new set below the selected set. The values are defaulted to zero.
- Copy – The Copy button copies the selected set as a new set in the list. The new set is placed directly below the selected set.
- Delete – The Delete button deletes the selected set. The rest of the sets are unaffected.
- Import – The Import button allows the user to select a file containing Incident Wave Conditions data. SMS prompts the user to replace or append to existing data. See the CGWAVE documentation for the Incident Wave Conditions file format.
- Clear – The Clear button deletes all of the Incident Wave Conditions data

Wave Condition Generation

The SMS interface for CGWAVE allows generation of multiple wave conditions from a spectrum in two ways. These are accessed using the buttons below the incident wave conditions.

- Generate From Spectrum – Wave conditions that make up a spectral wave condition can be added by selecting the spectrum to be simulated. A spectrum can be generated or read in using the Spectral Energy command CGWAVE menu.
- Long Wave Input Toolbox – The long wave input toolbox generates infra-gravity waves from a spectrum and input parameters.

CGWAVE Menu

The following menu commands are available in the CGWAVE Menu:

Command	Functionality
Spectral Energy	Brings up the spectral energy dialog to define/view wave energy spectra. This command also allows the user to visualize wave spectra.
Assign BC	This command is used to assign boundary conditions along a selected node string(s).
Material Properties	Material properties can be assigned and defined.
Model Check	Check for common problems.
Model Control	Brings up the Model Control dialog to specificy model parameters.
Reset 1D Spacing	...
Run CGWAVE	Brings up a dialog that allows the user to check what executable of CGWAVE should be run and then runs the model with the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

CGWAVE Model Checker

The CGWAVE model checker performs the generic mesh checking along with optionally checking to insure:

- that all boundaries mesh boundaries are assigned as land with a reflection coefficient or as open ocean.
- that all water depths are positive.

Related Topics

- CGWAVE Graphical Interface

CGWAVE Model Control

The CGWAVE model requires several user-specified parameters to control the analysis. The Model Control command from the CGWAVE Menu opens the Model Control dialog. This dialog contains parameters that control the execution of CGWAVE. The parameter description for each field is displayed in SMS using the interactive help messages.

Controls include:

- Title
- Incident Wave Conditions
- Open Boundary
- 1-D Options
- Solver
- Iteration Control
- General Parameters

Related Topics

- CGWAVE Graphical Interface

CGWAVE Numerical Solution

Equation (5) is generally solved using the boundary element method, the finite-difference method, or the finite element method. In general, finite-difference discretizations are not well-suited to represent the complex domain shapes described, for example, in Fig. 1. Not only are the boundaries distorted, but the number of uniformly spaced grids may also be excessively large. (Adequate resolution, typically 10 points per wavelength, demands that the spacing be determined from the smallest wavelength.) Most studies with the finite-difference method have been limited to largely rectangular domains (e.g. Li 1994a, 1994b; Panchang et al. 1991; Li and Anatasiou 1992). Boundary element models can handle arbitrary shapes and require minimal storage since only the boundaries are discretized; however, they are limited to subdomains with constant depths only (e.g. Isaacson and Qu 1990; Lee and Raichlen 1972; Lennon et al. 1982). Finite element models, on the other hand, allow the construction of grids with variable sizes (based on the local wavelength) and give a good reproduction of the boundary shapes. Most finite element models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Kostense et al. 1988; Demirbilek and Panchang 1998; Panchang et al. 2000) have used triangular elements, and modern graphical grid generating software permits efficient and accurate representation of harbors with complex shapes. The Surface Water Modeling System can be used to conveniently generate as many as 1,000,000 elements of varying size, based on the desired (user-specified) resolution, and to specify the desired reflection coefficients on various segments of the closed boundary. The solution of (1) by the finite element method is described in detail by Mei (1983) and by Demirbilek and Panchang (1998) when different types of open boundary conditions are used.

Whether one uses finite differences or finite elements for discretization, the numerical treatment of (1) with appropriately chosen boundary conditions leads to system of linear equations:

$$[A][\varphi]=[B] \quad (28)$$

where $[\varphi]$ represents the vector of all the unknown potentials. For solving (5), a similar system results as long as W is prespecified. The matrix [A] is usually extremely large. In earlier models (e.g. Tsay and Liu 1983; Tsay et al. 1989; Chen, 1990; Chen and Houston, 1987) the solution of (28) was accomplished by Gaussian Elimination, which requires enormous memory and is prohibitive when the number of wavelengths in the domain is large (i.e. short waves or a large domain). Pos and Kilner (1987) were able to alleviate this difficulty somewhat by using the frontal solution method of Irons (1970).

In recent years, the solution of (28) has been obtained with minimal storage requirements for [A]. This is due to the development by Panchang et al. (1991) and Li (1994a) of iterative techniques especially suited for (1). These techniques, based on the conjugate gradient method, guarantee convergence and have been found to be extremely robust in a wide variety of applications involving both finite differences and finite elements for several kinds of boundary conditions. For a review of other methods, see Panchang and Demirbilek (2000). Options based on the work of both papers, viz. Panchang et al. (1991) and Li (1994a), are available in CGWAVE. It is found that the latter often leads to faster convergence, but in an oscillating fashion. The former leads to a monotonically decreasing error which can be more reassuring while the iterations are in progress.

Related Topics

- CGWAVE
- Governing Equations
- Boundary Conditions

CGWAVE Practical Notes

Domain Shape: To the extent possible, it is necessary to have the coastlines outside the semicircle be as straight as possible, and ideally, parallel to one of the axis (as shown in Fig 1). The semicircle should be so located that the depths outside vary, to the extent possible, in a 1-d (cross-shore) direction only in the exterior. These depths are introduced in the model through the two 1-d sections. The user must specify a reflection coefficient for the exterior coastline.

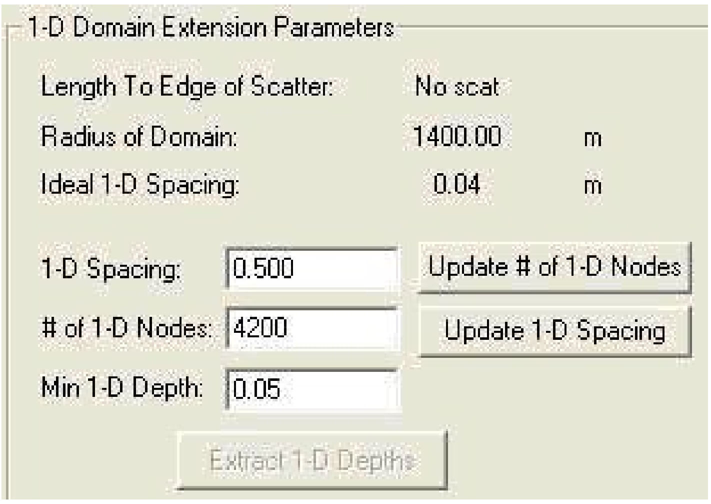
Of course, these conditions are not encountered in practice in an ideal manner, but it is important to understand that the model works under these assumptions. V-shaped coastlines do not meet these criteria. Also, not all real domains have 1-dimensionally varying exterior depths. Regions with 2-d depth variations should preferably be incorporated inside the semicircle and it may be necessary to enlarge the domain so that solutions in the area of interest (hopefully) are not affected by the coastline shape. It is impossible to accommodate all real topographic variations of the model's exterior domain. Since a part of what happens outside the semicircle influences the solution within (i.e. how the backscattered waves behave), some simplified representation of the exterior is needed. The exterior representation used in CGWAVE is more sophisticated than that used in earlier models (which assumed collinear and fully reflecting coastlines and constant exterior depths).

The size of the domain is also governed by the wavelength. Typically, the minimum radius for the semicircle should be about 2 or 3 times the wavelength, which also dictates the overall number of grids. It is important that the modeler first estimate the number of grid points needed, at least in an approximate sense. This can be done by estimating a nominal wavelength L (based on the period and depth), and using L/10 as a rough measure of the grid size. Obviously, large domains with short period waves will lead to hundreds of thousands of nodes. This is not a problem physically, but can be very time consuming to simulate. For spectral conditions, the domain size should be dictated by the longer wavelengths L_1 (say radius = $3L_1$), but the resolution for the shorter wavelengths (L_2) would be, say $L_2/10$, which may be excessively fine for the longer wavelengths. Obviously, some care (and perhaps compromise) is needed in designing the grid and also selecting the input spectral components. If some components have little energy, it may be best to delete them from the modeling.

The input wave condition (amplitude, period, direction) must be specified at the end of the 1-d cross-section. Simulations are less reliable (not necessarily wrong) if the waves approach the coast at a glancing angle (say within 20° of the coastline). The 1-d sections MUST reach out further than the outermost point of the semicircle. The domain should have no zero or negative depths; the land-water interface must be demarcated using a small water depth (say 0.5 m, but the limit depends on the overall water depths and wave periods and the problem being solved). Note that making this too shallow may result in a large number of grids in this area which may not be necessary for all applications).

For basic specifications, the incident wave angle specification is as follows. Waves going in the +x axis (i.e. to East) are specified as incident angle =0°. Waves going in +y direction (i.e. to North) are specified as incident angle =90°, to West =180°, to South = 270° . Incident wave direction is used in degrees. Convergence tolerance is typically assigned 10^{-8}, output frequency is 100 every iterations.

For the boundary condition specification, the "parabolic plus two 1d sections" option (Option 3 in .cgi) should be chosen for most harbor problems, with a semicircle open boundary; in earlier versions, it is possible to use the option with just one 1d section (Option 2 in .cgi) but it really provides no benefit over Option 3. The other options consist of the "parabolic approximation" (Option 1 in .cgi) which is suitable if the exterior region is of constant depth. The Fourier Bessel Series (Option 0 in .cgi) must be used if its limitations, described above in Section 2.2 (equations 15-17), are met; typically it is sufficient to specify 15 terms in the infinite series. For harbor problems, SMS will create the 1d sections based on depths based on depths on semicircle, and the desired grid spacing (used to solve the 1d equation (20)), based on (approximately) L/10 must be specified in the dialog box:

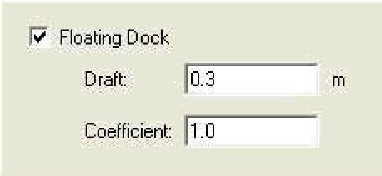


A full circle option is available for open sea problems where the exterior is of constant depth; the theoretical ideal for this is the Fourier Bessel Series (which works for unit wave input only); the parabolic mode may also be used (Panchang et al. 2000) with little loss of accuracy.

It is useful to perform initial sample runs, with no breaking and with fully absorbing coastlines, for a normally incident and obliquely incident wave input. The resulting solutions, especially the phase diagrams, can be used to perform visual quality control, since forward propagating waves are the easiest for making intuitive assessments (e.g. spacing between crests should be smaller and wave heights generally increasing in shallower water; bending of phase-lines for oblique incidence to approach the shore in a normal fashion, high wave heights in very shallow water because breaking was not applied, etc). Repeating the above runs with full reflection should lead to some standing wave patterns (large and small wave heights; rapid change of phase). A visual examination of the results will enhance confidence in later simulations.

There are two methods for applying wave breaking. For practical problems, we recommend making a non-breaking simulation and then applying an H/d limit (generally between 0.6 and 0.85) to alter the calculated wave heights to the limit (say 0.78d). The other, more rigorous approach is to run the model in the nonlinear breaking mode where breaking properties are continuously recalculated based on prior solutions. This requires several rounds of iterations and is recommended for special problems and/or research applications. For nonlinear runs, the maximum number of nonlinear iterations is generally specified as 15 with a tolerance of 10^{-6}. For spectral simulations, obviously the former approach is recommended.

For incorporating docks, the actual “without-dock” water depth is originally specified while generating the grid; areas covering the docks must then be highlighted, and a draft and coefficient, selected from Fig. 2 (Li et al. 2005) or Table 1, must then be specified for these areas. The coefficient = α = Aln(ka)+B. For kh < 0.1 and kh > 4, we recommend using the A and B values corresponding to these thresholds. For convenience, the numerical values corresponding to Figure 8 are given in Table 1. (Note, a = half width of the structure, relative submergence= draft/water-depth = d/h).



For shallow water, α is roughly equal to unity for shallow draft, and $\alpha \sim 0.7$ for deep draft. For intermediate and deep water, α is not constant but shows an increasing trend with ka. In very deep water, the mismatch is large. Note that for short waves (relative to submergence), T tends to 0. This requires us to create a high level of wave blockage, which can be accomplished by $\alpha \sim 0$ (Li et al. 2005). Of course this result would not hold if d/h were much smaller than the smallest value investigated by Li et al. (2005), which is 0.25 (typical in engineering practice).

It is worthwhile to refine the grid in these areas since the model uses the under-keel clearance multiplied by the coefficient as the depth for calculation. This modified depth is smaller than the original depth (and hence the resolution should be finer).

Finally, it is noted an extensive list of test-cases are provided in the appendix. CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. The user is highly encouraged to perform these simulations, alter parameters, etc. so that an examination of the results may help understand what can be expected. At a minimum, we recommend a visual inspection of the results provided. Often real-life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance the user’s confidence in his/her results, assuming the modeling was performed with due diligence. The test-cases can also be used as a teaching tool.

Return to CGWAVE Details

CGWAVE Test Cases

The following test cases are available with grids, input, and output files for the user to familiarize himself/herself with.

Test cases are available here with grids, input, and output files for the user to familiarize himself/herself. An extensive list of test-cases are also provided by the Coastal and Hydraulics Laboratory [3]. CGWAVE has been validated against these tests (which represents possibly the most rigorous testing for wave models). The model results are compared to lab data or analytical model results. The input and output files are provided. The user is highly encouraged to perform these simulations, alter parameters, etc., so that an examination of the results may help understand what can be expected. At a minimum, we recommend a visual inspection of the results provided. Often real life problems have complex solutions which are difficult to explain or even anticipate. The fact that the model reproduces the correct result in so many cases may enhance the user's confidence in his/her results, assuming the modeling was performed with due diligence. The test cases can also be used as a teaching tool.

The following tutorials may also be helpful for learning to use CGWAVE in SMS:

- General Section
  - Data Visualization
  - Mesh Editing
  - Observation
- Models Section
  - CGWAVE

Tests 1 & 2 [1]

These tests involve monochromatic wave propagation over the shoal-slope bathymetry of Berkhoff et al. (1982). Grid has 15 points per wavelength. Parabolic approximation open boundary condition. Test 1 – input amplitudelitude = 1 meter (linear). Test 2 – input amp = 0.0232 meter (nonlinear). Resulting amplification factors along Transect 5 are shown – they match results and data in Demirbilek and Panchang (1998). Wave direction and phases diagram shows largely progressive waves except near the shoal where the waves become multidirectional.

All runs involve no breaking.

Tests 3 & 4 [2]

Wave propagation over flat bottom and a shoal, after Vincent and Briggs (1989, JWPCOE). Monochromatic (T = 1.3 s) and broad-directional spectral (BI) input based on Panchang et al. (1990, JWPCOE). For spectral simulation, input consists of 29 directional components in the ±60° bandwidth and 5 frequency components. All runs involve no breaking. Results match numerical and experimental data described in Demirbilek and Panchang (1998), Vincent and Briggs (1989), and Panchang et al. (1990)

Test 5 [3]

Wave propagation (T=5.05 s, d = 0.25 meter) over a flat bottom surrounded by infinite ocean. **Depth = 0.25 meter**. Test 5 – using Bessel-Fourier boundary conditions (this is the most accurate boundary condition for the problem as specified although the exterior conditions are unrealistic in practice). See Xu et al. (1996, JWPCOE) for details. Note the Bessel-Fourier boundary condition works only for input amplitude = 1 meter. For other input amplitudes, solution should be appropriately scaled by user.

Test 6 [4]

As in test 5, but with friction which seems to become effective for long waves. Test 6 – the circular domain is assigned f = 0.5 everywhere, waves propagate in from the right. Friction leads to smaller wave heights.

Test 7 [5]

As in Test 5, but only the central area has a non-zero friction.

Test 8 [6]

Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used.

Test 9 [7]

As in Test 8, but with parabolic boundary condition only. This is used to demonstrate correctness of this boundary condition since solution for constant exterior depth is known.

Test 10 [8]

As in Test 8, but friction f = 0.5 for the whole domain. Note wave height input = 2 m. at the end of one-dimensional section which extends beyond the semicircle. Wave heights decrease in shoreward direction due to friction. No breaking used.

Test 11 [9]

As in Test 8, but with f = 0.5 in a central square region (can be seen on .cgi file). f = 0 elsewhere. No breaking.

Test 12 [10]

As in Test 10, but with central square area indicated as a “floating dock”.

Test 13 - Circular Island/Shoal [11]

Long wave propagation past the circular island/shoal combination of Homma (1950). Bessel-Fourier open boundary condition. Input T = 240 sec. Results match analytical solution given in Demirbilek & Panchang (1998). No breaking.

Test 14 [12]

As in Test 5, but with a circular pile in the domain. Input T = 10 s and constant depth = 15.03 meters. Results match analytical solution (see Panchang et al. 2000, ASCE JWPCOE).

Test 15 [13]

As in 14, but the pile is off-center. Parabolic open boundary condition. Results match analytical solution (Panchang et al. 2000, JWPCOE).

Test 16 [14]

Long wave (T = 260 s) propagation up a sloping beach. Parabolic and one-dimensional boundary condition. Solution is completely one-dimensional. Results match those in Panchang et al. (2000).

Test 17 [15]

Oblique wave incidence on uniformly sloping beach. Results match analytical solution of Radder (1979) given in Panchang et al. (2000, JWPCOE).

Test 18 [16]

Propagation of obliquely incident waves (incidence angle =20°) past a shore-perpendicular thin fully-reflecting breakwater on a sloping beach (beach is fully absorbing). Parabolic and one-dimensional open boundary condition. Results match analytical results given in Kirby (1986) and Panchang et al. (2000).

Tests 19 & 20 [17]

As in Test 18, but with nonlinear breaking on and off. Test pertains parameters in Zhao et al., (2000, Coastal Engineering).

Test 21 [18]

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for no breaking.

Test 22 [19]

As in Test 19, but with shore-parallel breakwater. Parameters and results as in Zhao et al. (2000). Results are for nonlinear breaking.

Test 23 [20]

Wave propagation/resonance in a rectangular harbor. Results match analytical solution plotted in Demirbilek & Panchang (1998). With friction f = 0.12, the resonant peak amplification reduces substantially for kl =1.4 (T = 1.0447 s).

Test 24 [21]

Wave propagation around a floating square platform in circular domain. While developing the 2-d grid, the area covering the dock is also filled with finite elements; each node is assigned a depth equal to the local under-keel clearance times the correction factor a. The parameters in the simulations are 2a = 2m, h = 1m, d/h = 0.5 and ka = 2 (corresponding to the cases described by Tsay and Liu (1983). So the depth used for calculation = α.d = α.(0.5) = 0.04. Correction factors such as a = 0.08 are given in Li et al. (2005, Canadian J. of Civil Engr). Results are similar to 3d results given in Tsay & Liu (1983).

Test 25 [22]

Wave propagation around a circular shoal in a circular domain. This is intended to show the effects of the "steep slope" terms. The test is based on Fig. 7 in Chandrasekhara and Cheung (1997, JWPCOE).

Test 26 [23]

Radiation Stress calculations. Waves propagating towards a coastline on a flat seabed, parabolic and one-dimensional open boundary condition. No breaking or friction used. Five folders are given. Three separate monochromatic cases (260 degrees, 270 degrees, and 280 degrees, i.e. normal incidence and 10 degrees off-center incidence) each of amplitude 0.5 m and period T = 1 s, for fully absorbing coastline. For 270 degrees, results for fully reflecting coastline are also given. Results match theoretical solution (eq. 6-9 and eq. 54 in Copeland, 1985, Coastal Engg).

For spectral tests, the same 3 waves were added to form the input spectrum. Radiation stresses for the spectrum are an integration of individual components (eq. 1 in Fedderson 2004, Coastal Engg). The spectral results can be used also to check the mean wave direction (which should be 270 degrees) and the mean frequency (= 6.28 radians/s).

Test 27 [24]

Wave propagates (incidence angle =0°) over a rectangular friction region in a constant-depth domain. With friction f = 2.122, input T = 20s, H = 6.1m and constant depth = 15.2m. Results match solution obtained by Dalrymple et al. (1984).

Test 28 [25]

Obliquely incident wave propagates in a rectangular channel with the fully-reflecting side walls. Input T = 12s and constant depth = 8m. Results match analytical solution plotted in Dalrymple and Martin (2000, JWPCOE).

Related Topics

- CGWAVE

References

[1] http://sms.aquaveo.com/CGWAVE-tests1and2.zip
[2] http://sms.aquaveo.com/CGWAVE-tests3and4.zip
[3] http://sms.aquaveo.com/CGWAVE-test05-flat.zip
[4] http://sms.aquaveo.com/CGWAVE-test06-flat(fr=0.5).zip
[5] http://sms.aquaveo.com/CGWAVE-test07-flat(fr=0.5c).zip
[6] http://sms.aquaveo.com/CGWAVE-test08-semi-cir%201d.zip
[7] http://sms.aquaveo.com/CGWAVE-test09-semi-cir.zip
[8] http://sms.aquaveo.com/CGWAVE-test10-semi-cir(fr=0.5)%201d.zip
[9] http://sms.aquaveo.com/CGWAVE-test11-semi-cir(fr=0.5cen)%201d.zip
[10] http://sms.aquaveo.com/CGWAVE-test12-semi-cir(fr-and-floating).zip
[11] http://sms.aquaveo.com/CGWAVE-test13-circ-island.zip
[12] http://sms.aquaveo.com/CGWAVE-test14-circ-cen.zip
[13] http://sms.aquaveo.com/CGWAVE-test15-circ-off.zip
[14] http://sms.aquaveo.com/CGWAVE-test16-slope1(f=1).zip
[15] http://sms.aquaveo.com/CGWAVE-test17-slope2(f=0).zip
[16] http://sms.aquaveo.com/CGWAVE-test18-slope-break.zip
[17] http://sms.aquaveo.com/CGWAVE-tests19and20.zip
[18] http://sms.aquaveo.com/CGWAVE-test21-akira_j(linear).zip
[19] http://sms.aquaveo.com/CGWAVE-test22-akira_j.zip
[20] http://sms.aquaveo.com/CGWAVE-test23-rect.zip
[21] http://sms.aquaveo.com/CGWAVE-test24-d1.0-c.zip
[22] http://sms.aquaveo.com/CGWAVE-test25-kh.zip
[23] http://sms.aquaveo.com/CGWAVE-test26-misc.zip
[24] http://sms.aquaveo.com/CGWAVE-test27.zip
[25] http://sms.aquaveo.com/CGWAVE-test28.zip

Long Wave Input Toolbox

Predicted infra-gravity (IG) wave input is required in modeling of long-waves affecting harbor. IG waves may also influence navigation, coastal inlets, and coastal structural design projects. To define wave conditions consistent with infra-gravity waves, the interface in SMS includes the infra-gravity wave input toolbox. This tool computes wave conditions for input to CGWAVE using a one dimensional Boussinesq analysis.

Input

The IGWT requests for the following inputs:

- An input wave spectrum (from file or from SMS) – This spectrum should define the wave conditions at the deep water point. SMS creates a series of frequencies and energy densities from this spectrum that are fed into the one dimensional Boussinesq model.
- Offshore water depth – This parameter refers to the depth in meters at the deepwater buoy (or site). If this is deeper than the Boussinesq limit, the depth will be set to the limit.
- Nearshore water depth – This refers to the depth in meters at the predominant breaking location. This value can be approximated as twice the incident significant wave height.
- Minimum/maximum long wave period – This refers to the cutoff periods for the IG wave spectrum. Typical values are from 30 to 600 sec.
- Number of components – using this parameter the user specifies how many wave components will be generated in each specified direction for CGWAVE. This term is referred to as 'N' in the discussion below.
- Maximum oblique angle – The one dimensional Boussineesq model ignores wave direction. In fact, the directional bins from the input spectrum are ignored as they are converted to a frequency spectrum. This value defines a total variation (in degrees) for the directions to be considered. They are centered around the shore normal direction.
- Number of angles – This value should be a positive integer. The toolbox will create N different wave components for each of this number of directions. If this number is 1, all the components will be generated in a shore normal direction. If this number is 2, then two sets of components will be generated, each half the oblique angle either side of the shore normal direction.

Output

The IGWT creates a series of wave components for each of the directions specified in the input. If the number of angles is 0, then there will be N components distributed through the frequency range. In the number of angles is 1, the toolbox will generate 3*N components (one direction offset from shore normal in each direction.

No check is made to remove zero energy components. The user should verify the generated components are what is wanted.

Approach

The IGWT utilizes a one-dimensional version of the BOUSS-2D model to transform wave spectrum from the "deep-water" limit of the Boussinesq model (H < L/2). A constant 1:50 slope is assumed between the offshore and nearshore water depths. If complex offshore topography exists, use BOUSS-2D to bring the waves to the nearshore.

External Links:

- May 2007 ERDC/CHL CHETN-I-73 May 2007 Infra-Gravity Wave Input Toolbox (IGWT): User's Guide [9]

Q&A CGWAVE

Return to the Main Q&A Page

Q: Where can I learn about the Infragravity Wave Tool box?

A: You can look at the Infra-Gravity Wave Input Toolbox (IGWT): User's Guide found on the ERDC website or by this link. IGWT [1]

References

[1] http://cirp.wes.army.mil/cirp/cetns/chetn-i-73.pdf

Saving CGWAVE

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.map referenced to new save location
- *.cgi referenced to new save location
- *.h5 referenced to new save location
- *.cgo referenced to old save location unless rerun then referenced to new save location

5.4. CMS

CMS

The Coastal Modeling System

The Coastal Modeling System (CMS) has been a research and development area of The Coastal Inlets Research Program (CIRP) at the United States Army Corps of Engineers – Engineering Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL) since 2006. It was build from a group of numerical models that have been under development since 2002. Information on the CIRP and publication on the CMS can be found at [1]

The system is a coordinated system of major multidimensional numerical models integrated to simulate waves, currents, water level, sediment transport, and morphology change in the coastal zone. Emphasis is on navigation channel performance and sediment exchanges between the inlet and adjacent beaches in the coastal zone. The CMS has been verified with field and laboratory data.

System Components

- CMS-Flow
- CMS-Wave

Steering

In order to combine the capabilities of the two main numeric engines of Flow and Waves, the user must pass information from one engine to the other. In the case of CMS-Flow, this means reading in wave data from CMS-Wave. Information passed to CMS-Flow includes reading radiation stress gradients that directly impact currents (wave driven currents) and height fields, wave directions and breaking data which enter into the sediment transport rate formulations. In the case of CMS-Wave, the option exists to read in currents and simulate their transformation by the current. For either situation, the data fields must be interpolated onto the native domain (interpolate wave data onto the flow grid and/or flow data onto the wave grid).

This may be done interactively using the tools in SMS. However, it is much more efficient to read or define simulations for both engines, and invoke the steering module from the Data menu. This tool runs the engines separately, but interpolates the output and passes it to the other engine automatically.

External Links:

- CIRP Wiki – online help database for CIRP information/publications and CMS modeling software [2]
- US Army Engineer Research and Development Center – Ongoing Research [3]
- Presentations
  - Coastal Modeling System (CMS) for Integrated Calculation of Waves, Flow, Sediment Transport, and Morphology Change [4]
  - Introduction to CMS-Wave [5]
  - Additional information on CMS-Flow capabilities [6]
  - CMS-Wave Demonstrations to Louisiana (Levees & Muddy coast) [7]

- Future of the CMS [8]

References

[1] http://cirp.usace.army.mil
[2] http://cirp.usace.army.mil/wiki
[3] http://www.erdc.usace.army.mil/pls/erdcpub/!www_fact_sheet.research_page?ps_rp_numb=118566&tmp_Main_Topic=&page=All
[4] http://cirp.usace.army.mil/workshops/mvn09/PDFs/5-IntroToCMS-SMS.pdf
[5] http://cirp.usace.army.mil/workshops/mvn09/PDFs/6-CMS-Wave-Intro.pdf
[6] http://cirp.usace.army.mil/workshops/mvn09/PDFs/7-CMS-Flow-modeling.pdf
[7] http://cirp.usace.army.mil/workshops/mvn09/PDFs/8-CMS-Wave-examples.pdf
[8] http://cirp.usace.army.mil/workshops/mvn09/PDFs/9-FutureOfCMS.pdf

CMS-Flow/CMS-Wave Steering

Steering CMS INLINE page – Provides the ability to perform CMS-FLOW ↔ CMS-WAVE steering with data stored in memory. The steering is controlled by the cms-flow model executable and not by SMS.

- CMS-FLOW Source Grid option – Select the cms-flow grid to be used in the steering.
- CMS-WAVE Source Grid option – Select the cms-wave grid to be used in the steering.
- Time options – Select the time parameters from the cms-flow model control interface.
- CMS-FLOW → CMS-WAVE options – Disabled. These options are set in the cms-flow model executable.
- CMS-WAVE → CMS-FLOW options – Disabled. These options are set in the cms-flow model executable.

Related Topics

- Steering
- ADCIRC / CMS-Wave steering

CMS-Flow/Save Points

Creating Save Points

Save points are cell-specific output that is used for CMS-FLOW simulations. To create save points do one of the following:

1. Create Save Points from an existing CMS_FLOW grid.
   - Click on the *Select Grid Cell* tool on the active grid.
   - Right click on one or more selected cells and select *Add Save Point(s)...*
     The CMS Save Point Manager is displayed showing the new cell(s). If no save points existed prior to this, a new save point coverage is created and a link will appear under the grid in the tree item data.
     – OR –
2. Right click on *Map Data* and select *New Coverage*. Find *CMS-Flow Save Points* under *CMS-FLOW* and click "Ok". A new save point coverage is created. Click on the new coverage so its active, then click the *Create Feature Point* tool. Click out some feature points. When finished, right click on the new coverage and select *Properties*. The *CMS Save Point Manager* is displayed showing the new cells.

Save Point Manager Dialog

The CMS Save Point Manager dialog is a spreadsheet showing individual save point cells. By default the "Name" column will display the grid cell id (if the save point was created from a grid). Attributes such as hydro, sediment salinity and waves and be turned on/off by toggling the values in the column. If the save points were loading into sms from a *.sp file, the cell row will be highlighted to indicate this.

Storing Save Points in the *.cmcards File

When saving the *.sms project. the save points get stored in the *.cmcards file. An example looks like this:

```
!Save Points
HYDRO_OUTPUT_INTERVAL       5.0 MINUTES
SEDIMENT_OUTPUT_INTERVAL    5.0 MINUTES
SALINITY_OUTPUT_INTERVAL    5.0 MINUTES
WAVE_OUTPUT_INTERVAL        5.0 MINUTES

SAVE_POINT "6, 61" -2867.5 2022.0 HYDRO SEDIMENT WAVE
SAVE_POINT "20, 59" -2517.5 1966.0 HYDRO
SAVE_POINT "10, 31" -2767.5 1182.0 SEDIMENT
SAVE_POINT "11, 31" -2742.5 1182.0 SEDIMENT
SAVE_POINT "10, 32" -2767.5 1210.0 SALINITY
SAVE_POINT "11, 32" -2742.5 1210.0 WAVE
```

The SAVE_POINT is formatted [name][x location][y location][hydro (if on)][sediment (if on)][salinity(if on)][wave(if on)]

.sp/.spx File for Reading in Save Point Information

Simulation output data gets stored in *sp and *.spx file. The *.sp file is and individual output file. The 8.spx file simply identifies all of the *.sp files that should be loaded for a simulation. An *.sp file looks something like this:

```
SAVE_POINT_OUTPUT  eta
REFERENCE_TIME     2001/01/01 00:00 -0000 GMT
CREATION_DATE      2012/05/28 09:02 -0600 GMT
CMS_VERSION        4.00.11
TIME_UNITS         HOURS
OUTPUT_UNITS       'm'
NUMBER_POINTS      2

        NAME_BEGIN
            '6, 61'
            '20, 59'
        NAME_END

     XY_BEGIN
        -2867.5000  2022.0000
        -2517.5000  1966.0000
     XY_END

   SCALAR_TS_BEGIN
      0.0000  0.0000E+00  0.0000E+00
      0.0333  1.7545E-09  1.7434E-09
      0.0667  1.4042E-08  1.3971E-08
      0.1000  4.6679E-08  4.6524E-08
      0.1333  1.1143E-07  1.1117E-07
```

Sms will read in a *.sp file and create a cms-flow save point coverage with the points.

Time Series Plot Wizard

*.sp files that are read into sms can be plotted into a time-series curve. To view the curve do the following:

1. Load a *.sp file into SMS
2. Right-click on *Display→Plot Wizard* from the menu dialog. Select *CMS Save Points* from the plot type then click next. Change the coverage to *Specified object(s)*. Now select the points you wish to plot. Now select the *Solutions and Datasets* then click "Ok". A time series plot is created.

5.4.a. CMS-Flow

CMS-Flow

CMS-Flow	
Model Info	
Model type	Hydrodynamic model intended for local applications, primarily at inlets, the nearshore, and bays
Developer	Christopher W. Reed, Ph.D. Alejandro Sanchez Mitchell E. Brown
Web site	[1]
Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Flow

CMS-Flow is a component of the Coastal Modeling System (CMS). Until 2007, it was developed under the name M2D. At that point in time, it was revised, file formats were updated for better flexibility and expandability, and it was incorporated into the CMS suite.

The model developers at the United States Army Corps of Engineers maintain a wiki specifically for the numerical engine. It can be viewed at: cirp.usace.army.mil/wiki/CMS-Flow [1]. For more information on the model itself, refer to the users manual [14] published by USACE-ERDC.

CMS-Flow is a finite-volume numerical engine which includes the capabilities to compute both hydrodynamics (water levels and current flow values under any combination of tide, wind, surge, waves and river flow) sediment transport as bedload, suspended load, and total load, and morphology change.

The interface in SMS allows the user to set up and edit computational grids, specify model parameters, define interaction of this model with the wave counterpart (CMS-Wave), launch the model and visualize the results.

The model is intended to be run on a project-scale, meaning the domain should only be on the order of 1-100 kilometers in length and width. The following sections describe the interface and make recommendations for application of the model.

Graphical Interface

The CMS-Flow Graphical Interface is contained in the Cartesian Grid Module as well as the Map Module and includes tools to create and edit a CMS-Flow simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to CMS-Flow. If a grid has already been created for a CMS-Flow simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to CMS-Flow. See Creating 2D Cartesian Grids for more information.

Using the Model / Practical Notes

For new simulations, users will create the CMS-Flow grid based on a conceptual model. The conceptual model includes:

- Grid Generation – We recommend that you generate a CMS-Flow grid using the conceptual model and a CMS-Flow Coverage. This coverage has attributes associated with a two-dimensional Cartesian grid and the model parameters associated with CMS-Flow. The grid position and extents are defined in the coverage using a grid frame, which you can define with three clicks of the mouse (recommendation is to click the lower left corner, lower right corner and then upper right corner, but the position, orientation and size can all be edited during the grid generation process. The coverage also defines the location of land and water in the grid using one of three methods:
    - Land/Water cells defined based bathymetric values – CMS-Flow uses depths, so positive depth indicates water, negative depth indicates land. Cells with depth less than the negative value of the water surface are dry. This option requires a geometric survey that includes both the bathymetric area and the areas that could potentially be flooded. This is the most intuitive option and the preferred method if geometric data is available.
    - Land/Water interface defined by coastline arcs – This option allows the user to define, read or import arc definitions that delineate the water area. These arcs include an orientation. To the left of the arc is land, to the right is water. The user can select an arc and swap its orientation. All the area inside the grid frame on the "water" side of the arc must have elevations defined either from a survey, or by specification. Cells created on the "land" side of the arc will never be included in calculations (they are permanently dry). These arcs also include an attribute defining how cells spanning this interface are to be classified. They may be forced to be water (ocean preference), forced to be land (land preference) or split based on the percentage of the cell on each side of the arc (percent preference).
    - Land/Water interface defined by polygons – This option also requires the user to define arcs delineating the extents of the computational area. However, these arcs must be closed into polygons. Each polygon is specified to enclose land or water and cells are classified accordingly.
- Model Output – The numerical engine consists of several components. The base engine computes hydrodynamics. To this, sediment transport and salinity can be enabled as well. Each process produces spatially varied solutions (values for each wet cell) that SMS can display as spatial datasets. Additional observation cells can be created to view output at a higher temporal resolution.

Case Studies / Sample Problems

The following tutorials may be helpful for learning to use CMS-Flow in SMS:

- Models Section
    - CMS – CMS-Flow

Related Links

- SMS Models page
- CMS-Wave

External Links

- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [8]
- Aug 2006 ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [14]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [18] Updated FAQ Website [2]
- May 2005 ERDC/CHL CHETN-IV-63 Representation of Nonerodible (Hard) Bottom in Two-Dimensional Morphology Change Models [27]
- May 2004 ERDC/CHL TR-04-2 Two-Dimensional Depth-Averaged Circulation Model M2D: Version 2.0, Report 1, Technical Documentation and User's Guide [30]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D and STWAVE [32]

References

[1] http://cirp.usace.army.mil/wiki/CMS-Flow
[2] http://cirp.usace.army.mil/pubs/FAQs/FAQ.html

CMS-Flow Cell Attributes

Individual cells of a CMS-Flow grid can be of four attributes which define the cells' computational inclusion into the simulation and controls specific computation output.

The cell attributes are as follows:

- Inactive land – This cell only exists within SMS, it will not be saved as part of the CMS-Flow computational domain. If this cell is adjacent to an active ocean cell, then interface between the two cells will be treated as a wall.
- Active ocean – This is a computational cell.
- Observation – This is a computational cell which also represents an observation station. CMS-Flow can output calculations of time series (water elevation and velocities), flow rate and transport at the cell.

This window is accessed from the *CMS-Flow | Assign Cell Attributes...* menu item or the *Cell Attributes...* right click menu associated with the *Select Grid Cell* tool with at least one cell selected.

Dialog Description

This dialog mainly consists of the *Inactive land*, *Active ocean*, *Inactive ocean*, or *Observation* radio group. Only one cell type may be chosen. If *Observation* option is selected then the observation output type check boxes and *Identifier* edit field are enabled. At least one output type must be selected for a valid observation station.

If multiple cells with differing cell types are selected, the window will not display a selected cell type and the *OK* button will be disabled. This is to protect the previous assignment from being overwritten by mistake. The *OK* button will be enabled when an option is selected. If multiple cells with differing observation output types are selected, the window will show *Observation* selected, the differing output type check boxes will be in a tri-state mode (checked with the check mark dimmed) and the *OK* button will be enabled. This is to represent that some cells are of the type and the rest are not. By clicking on the check box in tri-state mode, the new assignment (on or off) will be applied to all selected cells when *OK* is selected. A check box cannot be returned to tri-state mode once changed, select *Cancel* to omit assignment. A check box in tri-state mode when *OK* is selected means that that specific output type will not be changed, though other assignments (fully checked check boxes or identifier text) may be applied. The *Identifier* edit field works similarly, however, a note ("*Not the same for all cells") will appear when the identifying text is not unique for all selected cells. In this case, the edit field will display the last non-blank identifier found while checking the selected cells. If the note is present, then assignment of identifier text will not occur. The note will disappear if the text is changed, and therefore the text assignment will be applied to all selected cells when *OK* is selected.

Related Topics

- CMS-Flow

CMS-Flow Graphical Interface

The CMS-Flow graphical interface includes tools to assist with creating, editing, and debugging a CMS-Flow model. The CMS-Flow interface exists in the Cartesian Grid Module. The Map Module also provides flexibility when creating the grid.

CMS-Flow Menu

The interface consists of tools to create the grid, a CMS-Flow menu with model specific commands and visualization tools for analyzing the datasets computed by the model. The menu includes the following commands:

- **Assign BC...** (Boundary Condition) – This command allows the user to assign a boundary condition to the cell string through the CMS-Flow Boundary Conditions Dialog.
- **Delete BC...** – This command allows the user to delete the boundary condition assigned to that string.
- **Cell Attributes...** – This command allows the user to set an individual cells status.
- **Merge Cells** – This command allows the user to merge consecutive rows or columns into one.
- **Model Check...** – The CMS-Flow Model Check performs a number of checks on the simulation to ensure a valid model.
- **Model Control...** – This command allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation.
- **Run CMS-Flow...** – This command launches the CMS-Flow model for the current simulation.

See CMS-Flow Menu for additional information.

Setting Up/Editing a Model

Grid Creation

A grid for CMS-Flow consists of varying sized rectangular cells. Each cell defines the depth from a fixed datum to the bottom of the water column at the centroid of the cell. CMS-Flow computes the water level at the cell centroid, and the flow velocity across each cell face. A grid may be created in one of two ways including:

- Through the Cartesian Grid module – This option allows for the creation of a simple grid consisting of cells of uniform size (they may be rectangular, but all the same size). See Creating 2D Cartesian Grids for more information. This grid may be edited in the Cartesian Grid module (see Editing 2D Grids).
- Using a CMS-Flow Coverage in the Map Module – This method results in a grid that can be edited with all of the tools available in the Cartesian Grid module, but also allows flexibility in two ways:

- Refine points may be created to specify cell resolution in one or both directions at specific locations inside the domain. This allows areas of high concern, and high variability, such as in the mouth of an inlet to be defined with specified high resolution, which transitions to lower resolution in areas away from these regions.
- The grid frame, with the grid definition, is saved and can then be copied and edited to create variations of grid resolution and coverage.

Boundary Conditions

All numeric models require boundary condition data. In CMS-Flow, boundary conditions are defined on cell strings.

Cell Strings

For CMS-Flow, a cell string consists of a series of contiguous or adjacent cells. This means the cells share at least a corner and usually an edge. When SMS creates a grid, it also creates cell strings around the computational domain. This consists of open ocean cells along the edge of the domain as well as the last ocean cells adjacent to the land. SMS will create one cell string for each continuous section of ocean along the grid boundary, one cell string for each section of coastline, and one cell string for each island. Each cell string may be assigned an attribute to describe the type of boundary condition applied at that location. If no specification is made, the default boundary condition is a closed boundary (no flow is allowed across this boundary). This is typically the case for coastline and island cell strings. These cell strings are actually not even required for CMS-Flow, but are useful when using CMS-Flow output in a Particle Tracking Model (PTM) simulation. Cell strings may be split into multiple cell strings by selecting the cell where the break is desired and selecting the Split command from the Cellstring menu. Cell strings may be merged by selecting two (or more) cellstrings that connect in a continuous line, and selecting the *Merge* command from the Cellstring menu.

Assigning Boundary Conditions

The user assigns a boundary condition by activating the *Select Cellstring* tool, selecting the cell string, and then selecting the *Assign BC...* command from the menu. This invokes the *CMS-Flow Boundary Conditions* dialog and allows the user to define the conditions along the cell string. Multiple cell strings may be selected and assigned at the same time, but this is not common. Typically, the boundary conditions will only be applied to cell strings along the edge of the domain, but may be applied to cell strings on the interior if a portion of the domain is to be excluded from computation (such as cutting off a portion of a bay) for computational efficiency. Boundary condition types supported by CMS include:

- Land – This is the default condition and indicates that no flow will cross this boundary.
- Flow rate-forcing – This allows the user to specify an inflow rate (flow in cubic meters/second at each cell). This can be used to represent a river flowing into the domain.
- WSE-forcing – This allows the user to specify the water surface elevation as a function of time for the cellstring. Options include specifying a single curve (water level -vs- time) and all the cells will have the same water level at the specified time and extracting individual curves for each cell either from a regional tidal database (ADCIRC database) or from a regional (larger) circulation model.
- Tidal constituent-forcing – This allows the user to specify the water surface as a function of the principal tidal constituents around the domain. When this option is selected, the user must specify which constituents (along with their amplitudes and phases) in the Model Control dialog. All cells in the tidal cell strings will be forced with the same water levels.
- WSE and Velocity-forcing – This option allows the user to extract both water levels and velocity values for each cell in the cellstring from a regional model. This requires a regional model to extract from.

Cell Attributes

The user may select cells and assign attributes to them using *Select Cell* tool and two commands from the CMS-Flow menu.

If a cell is selected, its bathymetric depth may be edited using the "Z" edit field at the top of the screen.

For any selected cells, the *Assign Cell Attributes* command allows the user specify the cell as water or land, and if it is water, specify that it is an observation cell. This option instructs the model to output data at a higher frequency at this location for analysis after the simulation is complete.

When entire adjacent rows or columns are selected, they may be merged to decrease resolution in this area of the domain. (Warning, this operation deletes all cell strings, so all boundary condition specification would be lost if this operation is performed. Make sure you do this type of operation during grid construction.)

Model Check

Whenever the user issues the *Model Check* command, and just before launching an analysis, SMS performs a check on the model. If conditions are detected that are out of the ordinary or recommended range, warnings are given with suggestions to correct the problem. If serious problems are detected, errors are reported, again with suggestions.

Model checks include checking on the aspect ratio of the cells, ensuring that compatible boundary conditions are specified, and ensuring that model parameters are within acceptable ranges.

The model check is not all inclusive. Just because a model passes the model check does not guarantee the model will run and produce viable solutions.

Model Control

The CMS-Flow Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The CMS-Flow files are written automatically with the SMS project file or can be saved separately using the *File | Save CMS-Flow* or *File | Save As* menu commands. See the CMS-Flow publications from ERDC for more information on the files used for the CMS-Flow run.

CMS-Flow can be launched from SMS using the *CMS-Flow | Run CMS-Flow* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *CMS-Flow | Model Check* menu command.

Related Topics

- Hard Bottom
- Observation Cells
- Cell Attributes
- Advanced Cards
- Salinity
- Create a 2D Cartesian grid

CMS-Flow Grid Modification Flag

As CMS-Flow runs, especially when coupled with CMS-Wave, water levels increase and decrease and bed morphology changes the bottom elevations. Some cells may change state from dry to active. When this occurs, the datasets associate with grid change. The grid modification flag was created to indicate that the grid status has changed so that the model knows that geometric relationships (such as neighbor files) need to be updated.

The grid modification flag will start at 0 and increment by 1 when one of the following occurs:

- Merging Rows and columns
- Splitting rows and columns
- Dragging row or column edges
- Duplicating a grid
- Making an active ocean cell inactive or vice versa

The flag is written out to the *.cmcards file with the GRID_MODIFICATION_NUMBER card. Below is an example:

GRID_MODIFICATION_NUMBER 4

Related Topics

- CMS-Flow

CMS-Flow Hard Bottom

CMS-Flow includes a Hard Bottom morphologic constraint that "provides the capability to simulate mixed bottom types within a single simulation," (ERDC/CHL TR-06-9 Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change [14], p. 33). Hard bottom is a cell depth constraint applied to areas of the grid which represent exposed or covered non-erodible material such as bedrock. This specification limits the erosion of sediment to the hard bottom depth and therefore constrains water depth. During sediment transport calculations, exposed hard bottom cells may become covered through deposition.

By default, CMS-Flow cells are fully-erodible cells with no specified hard bottom depth (inactive cells; denoted by the CMS-Flow null value of -999.0).

When specified, cell hard bottom depths will appear in the Project Explorer as a scalar dataset beneath the CMS-Flow grid. This dataset can not be deleted, though it can be edited like any other dataset. A CMS-Flow simulation must contain the hard bottom dataset (even if it is not specified) so SMS will create a defaulted (inactive cells) dataset if it does not already exist when saving the simulation.

Hard bottom should be specified only for computational (ocean) cells.

The hard bottom dataset can created, edited, viewed and verified using the following SMS interface features.

Model Control

Within the CMS-Flow Model Control window, the hard bottom dataset can be created from the *Sediment* tab. If the dataset does not exist, it can be created using the *Create Dataset* button. If a dataset exists (created using the Data Calculator) which represents the intended hard bottom specifications, the *Select Dataset...* button can be used to select such dataset and copy the values to the hard bottom dataset.

Hard Bottom Specification

Although the hard bottom dataset can be edited (when its the active dataset) by selecting a cell (or group of cells) and changing the scalar (S) value in the *Edit Window*, an user-friendly window exists which provides specification options. With the *Select Grid Cell* tool active, make a selection, right click to bring up the tool menu and choose the *Specify Hard Bottom...* option. This will open the *CMS-Flow Hard Bottom Specification* window.

The following options are provided in the *Hard Bottom Specification* window:

- **Use bathymetric cell depth –** Sets the cell hard bottom depth to be the cell geometry value thereby creating an exposed non-erodible condition. If multiple cells were selected, then each cell will use its respective bathymetric depth.
- **Specified distance below bathymetric cell depth –** Sets the cell hard bottom depth to be the cell geometry value plus the specified distance thereby creating a sediment-covered non-erodible condition. The distance is limited to positive values to ensure the hard bottom depth is greater than the geometry value. The cell can provide sediment for transportation, however, the amount of erosion is limited. If multiple cells were selected, then each cell will use its respective bathymetric depth.
- **Specified depth –** Sets the cell hard bottom depth to the specified depth thereby creating a sediment-covered non-erodible condition similar to specified distance. The depth is limited to greater than the geometry value. If multiple cells were selected, then the depth is limited to greater than the largest geometry value and all cells will have the same value.
- **Unspecified –** Resets to an inactive hard bottom condition. The cell hard bottom depth is set to the CMS-Flow null value. If multiple cells were selected, then all cells will be reset.

If no cells are selected when opening the Hard Bottom Specification window, then all computational (ocean) cells will be used. If a selection of only non-computational cells, then specification cannot occur. If a selection contains computational and non-computational cells, then the specification will only apply to the computational cells.

If multiple computational cells with differing specifications are selected, the window will not display a selected specification type and the *OK* button will be disabled. This is to protect the previous specifications from being overwritten by mistake. The *OK* button will be enabled when an option is selected. The minimum hard bottom depth of the multiple computational cells selected will be displayed in the *Depth* edit field and the minimum hard bottom depth minus the maximum geometry depth of the multiple computational cells selected will be displayed in the *Distance* edit field.

Display Options

The hard bottom dataset (when its the active dataset) will only display the cells with hard bottom specified if the *Ocean cell* display option is turned on. Inactive hard bottom cells are not displayed.

CMS-Flow includes hard bottom symbols to differentiate specifications. On the *Cartesian Grid* page of the Display Options window (when CMS-Flow is the active model), the *Hard bottom symbols* check box controls the display of symbols that will appear in hard bottom cells (even if the hard bottom dataset is not active). If this is turned on, then the user must be aware of the individual symbol settings accessed by clicking on the *Options...* button. The *Options...* button displays the CMS-Flow Hard Bottom Symbols window.

Hard bottom symbols can be selected for three hard bottom specification types:

- **Non-erodible** – Displayed in exposed hard bottom cells (cell hard bottom depth is equal to cell bathymetric depth).
- **Erodible to specified depth** – Displayed in sediment-covered hard bottom cells (cell hard bottom depth is greater than cell bathymetric depth).
- **Invalid specification** – Displayed in hard bottom cells where the hard bottom depth is less than cell bathymetric depth (the geometry is below the erosion limit).

If the *Hard bottom symbols* check box is turned off, no symbols will be displayed and the individual settings cannot be accessed, however, the individual settings will not be changed.

Model Check

The CMS-Flow Model Checker, accessed from the CMS-Flow | Model Check... menu item, includes a check to ensure that no invalid hard bottom specifications exist in the grid. An invalid specification may be created, for example, by setting an infeasible hard bottom scalar value in the Edit Window or adjusted the grid's geometry without updating the hard bottom. It is suggested that the model checker be used prior to running CMS-Flow.

Related Topics

- CMS-Flow

CMS-Flow Hot Start

In the CMS-Flow Model Control, the user can specify a previously saved hot start file to be used as initial conditions or instruct CMS-Flow to save hot start files for future use.

To create a hot start file, either select *Write Hot Start output file* and select an output time, or select *Automatic recurring Hot Start file* and choose an output interval.

- Choosing to write the hot start file at a specific output time will create the following file:
  - *hot_start.h5* This file has the simulation data including elevations and velocities.
- Choosing automatic recurring hot start files will create the following files:
  - *HOTSTART.INFO* This file records what time the hot start file was written and which hot start file is the most recent.
  - *HOTSTARTx.H5* (where "x" is a counter) These files have the simulation data including elevations and velocities.

Once the hot start files are created, they can be read into CMS-Flow. Open the CMS-Flow Model Control and check the "initial conditions file" check box, then select the hot start file you wish to use.

When using a hot start file, the following parameters should be changed as follows:

- *Start Date*: no change
- *Start Time*: no change
- *Simulation Duration*: decrease by the duration of the hot start file
- *Boundary Conditions*: no change

Note: For a simulation using a hot start file, the first time step of the solution will be the start time plus the value of one time step plus the duration of the hot start file.

Related Topics

- CMS-Flow

CMS-Flow Menu

The CMS-Flow menu includes commands for editing and running a simulation. This includes viewing and assigning boundary conditions and cell attributes, checking the simulation for common errors, and specifying model parameters. Commands from the menu include:

- **Assign BC... –** This command is inactive unless a cell string is selected. It allows the user to assign a boundary condition to the cell string through the CMS-Flow Boundary Conditions Dialog. Depending on the boundary condition type, curves may be created or extracted to define the boundary condition. When appropriate, the file browse button allows the user to load an already-existing time series for use with one of the time series boundary conditions. Boundary conditions specified within this dialog include:
  - **Land –** When a cell string is created, the string receives this assignment. This defines a boundary as closed and no flow will cross this boundary.
  - **Flow Rate-forcing –** A time series must be defined (by clicking the Define Curve… button) that contains a list of times and flow rates in m3/sec along the boundary.
  - **WSE-forcing –** A time series must be defined for either the string or each cell in the string. The time series associated with this boundary contains a list of times and water levels in meters along the boundary. An option to extract time series curves or each cell can be applied to regional models loaded into SMS or create time series curves from tidal constituents.
  - **Tidal Constituent-forcing** Uses up to eight tidal constituents to force with. Tidal constituents are specified within the CMS Tidal Constituent dialog accessed by clicking "Global Attributes.." in the "Options" section of the Boundary Conditions dialog. The CMS-Flow Tidal Constituents dialog allows the user to specify the tidal constituents to force with along any tidal constituent-forcing boundaries that have been defined within the M2D grid. The user must specify an amplitude and phase for each constituent that will be used to force with. In SMS 11.1 Up to 37 constituents may be used: M2, S2, N2, K1, M4, O1, M6, MK3, S4, MN4, NU2, S6, MU2, 2N2, OO1, LAM2, S1, M1, J1, MM, SSA, SA, SF, MF, RHO, Q1, T2, R2, 2Q1, P1, 2SM2, M3, L2, 2MK3, K2, M8, MS4. Earlier versions us up to eight constituents may be used: M2, S2, N2, K2, K1, O1, M4, and M6.
  - **WSE and Velocity-forcing –** This option adds the requirement to specify currents at each cell to the WSE-forcing option. These must be extracted from a regional model.
  - **Harmonic WSE-forcing –** With this condition, the user can specify the components of the forcing curve after selecting "Global Attributes..." in the Options section of the Boundary Conditions dialog. The specification of terms of the time series consist of amplitude (m), frequency (cycles/hr) and phase (deg). Values are written out in the *.cmcards.file with the HARMONIC_BEGIN and HARMINIC_END cards. Assigned cellstrings are written as HARMONIC_CELLSTRING in the same file.
- **Delete BC –** This command is inactive unless a cell string is selected. It allows the user to delete the boundary condition assigned to that string.
- **Assign Cell Attributes –** This command allows the user to set an individual cells status. A cell can be active (water), inactive (land), or observation. Observation cells are assumed to be active. The user also specifies a roughness value for the cell. In future versions a material type will be specified instead of a roughness value directly. Observation cells can be assigned where CMS-Flow can write time series and/or flow rate output files.
- **Merge Cells –** This command allows the user to merge consecutive rows or columns into one. The select row or column tools must be used to perform this operation.

- **Model Check...**

  The CMS-Flow Model Check performs a number of checks on the simulation to ensure a valid model. These checks include:

  - All water boundaries have boundary conditions specified.
  - Cell aspect ratio is no greater than 2.
  - Max time step size is suggested according to an internal calculation made within SMS.
  - Make sure model control has been set up correctly (i.e. if a tidal boundary exists, make sure that tidal constituents have been defined to force with).
  - Ensure that CMS-Flow has been set up to output the specific data for any existing observation station and that observation stations exist for any specified output data. Observation is not required to run CMS-Flow so this is a reminder that something may have been intended but forgotten.
  - Ensure that no invalid hard bottom specifications exist in the grid. An invalid specification may be created, for example, by setting an infeasible hard bottom scalar value in the Edit Window or adjusted the grid's geometry without updating the hard bottom.

  The CMS-Flow Model Checker is accessed from the CMS-Flow | Model Check... menu item. It is suggested that the model checker be used prior to running CMS-Flow.

- **CMS-Flow Model Control...** – The *CMS-Flow Model Control* dialog allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes six tabs which partition the parameters into related groups. The tabs and their related parameters include:
  - General – General model parameters and options.
  - Flow – General model parameters and options.
  - Wind – Wind datasets. These are optional spatially varied input to the engine.
  - Wave – Wave datasets. These are optional spatially varied input to the engine.
  - Output – Parameters associated with global output.
  - Advanced – This tab allows those working with model developers to work with experimental options in the model.
- **Run CMS-Flow ...** – This command launches the CMS-Flow model for the current simulation.

Related Topics

- CMS-Flow

CMS-Flow Model Control

The CMS-Flow Model Control dialog allows the user to view and edit the current parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes six tabs which partition the parameters into related groups. The tabs and their related parameters include:

General

This dialog allows the user to specify general flow model parameters, controlling which model options the simulation will employ. The controls include:

Time Control

In this section the user can set the starting time, duration and hydraulic time step. The Ramp duration defines the length of an incremental loading portion at the beginning of the simulation. Controls include:

- Start date and time
- Simulation duration or end date and time
- Ramp duration

Hot Start

These controls allow the user to specify a previously saved hot start file to be used as initial conditions or instruct CMS-Flow to save hot start files for future use.

- Initial conditions file
- Write Hot Start output file
- Time at which Hot Start output file should be written
- Automatic recurring Hot Start file
- Interval for writing recurring Hot Start file

Threads

CMS-Flow can take advantage of multiple processors using Open-MP parallelization technology. This control allows the user to specify the maximum number of threads that the engine should occupy.

Solution Scheme

CMS-Flow now has two solution schemes. These include the traditional "explicit" scheme. This method requires that flow be tracked through each element, resulting in hydrodynamic time steps in the order of one to two seconds. The new option is an "implicit" scheme which uses finite volume technology, supports much larger time steps resulting in fewer time steps (shorter run times) and is identically mass conserving. When the implicit scheme is being used, one of the following matrix solver types should be specified.

- GMRES
- GAUSS-SEIDEL
- GAUSS-SEIDEL-SOR
- BICGSTAB

Flow

This dialog allows the user to specify general hydrodynamic or flow model parameters, controlling which model options the simulation will employ. The controls include:

Hydrodynamic Time Step

This control allows the user to set the hydrodynamic time step for the model. If the explicit solution scheme is being used, a recommended timestep will be calculated based on grid cell size and depths.

Water Parameters

This section allows the user to specify various general parameters to be used by the simulation. This include:

- Water temperature
- Water density
- Depth to begin drying cells: This is a minimum depth of water before the model considers the cell to be dry. Shutting off cells when the depth drops below a tolerance avoids flutter of insignificant transfers of water. The recommended default value is 0.05 meters.
- Include advective terms
- Include mixing terms
- Include wall friction: Normally friction at the closed boundaries are ignored. The model can be instructed to include friction along these boundaries as if a wall exists there.
- Include stokes velocities

Bottom Friction Dataset

Friction datasets can be created and specified to define a spatially variable bottom roughness to provide a resistance to flow. This value has little impact in deep ocean applications but can be important in shallow regions. This is a user editable dataset. The available dataset types are:

- Mannings N
- Bottom friction coefficient
- Roughness height

A bed-slope friction coefficient toggle gives the option to specify a wave-current bottom friction type and a coefficient.

Turbulence Parameters

CMS-Flow now supports turbulence parameters.

- Model: Subgrid, Falconer, Parabolic, Mixing length
- Base value
- Current bottom coefficient
- Current horizontal coefficient
- Wave bottom coefficient
- Wave breaking coefficient

Wind

This dialog allows the user to specify wind conditions. These are optional inputs to the engine.

Wind Data

The user can turn on the option to tell CMS to include wind calculations in the simulation. Wind is simulated in CMS-Flow as a spatially constant, but temporally varying quantity. The model does not currently support tropical cyclonic winds.

- Spatially constant
  - Import from File...: Button allowing the user to import wind data from a file. The file must be in an "m2w" wind format.
  - Velocity (m/s): Button which invokes a time series editor to specify the variation of wind velocity through time in the simulation.
  - Direction (deg): Button which invokes a time series editor to specify the variation in direction of the wind through time in the simulation.
  - Anemometer height (m)
- Meteorologic stations - This option allows the user to insert data for multiple locations
  - Import from File...: Button allowing the user to import wind data from a file. The file must be in an "m2w" wind format.
  - Coordinates (X,Y): Location of meteorologic station
  - Velocity (m/s): Button which invokes a time series editor to specify the variation of wind velocity through time in the simulation.
  - Direction (deg): Button which invokes a time series editor to specify the variation in direction of the wind through time in the simulation.
  - Anemometer height (m)
- Temporally and spatially varying from file
  - Navy fleet numeric with pressure
  - OWI/PBL
  - Single ASCII file

Navy fleet and ASCII files require the following input:

- Number of X values
- Number of Y values
- Minimum X location
- Maximum Y location
- Time increment
- Distance between X values
- Distance between Y values

OWI/PBL requires three different files:

- Oceanweather Wind File
- Oceanweather Pressure File
- Oceanweather XY File

For more wind file information, see: http:/ / cirp. usace. army. mil/ wiki/ CMS-Flow_Wind_Forcing#Spatially_Variable_Wind_and_Atmospheric_Pressure [1].

Wave

This dialog allows the user to specify wave conditions. These are optional inputs to the engine.

Wave Data

CMS-Flow includes three options on how to use wave data.

- None – no wave data is imputed into the model.
- Single wave condition – the effects of waves are input to the model in the form of spatially varying datasets. These datasets must already exist on the grid. The user simply selects which dataset to use
  - Height
  - Period
  - Direction
  - Dissipation
  - Stress gradients (vector dataset)
  - Surface roller stress gradients (optional)
- Inline steering
  - CMS-Wave file. Location of the grid file or the simulation file
  - Steering Interval or automatic value
  - Specify extrapolation distances. If checked this will write out FLOW_EXTRAPOLATION_DISTANCE and WAVE_EXTRAPOLATION_DISTANCE cards.
  - Wave Water Level – Option to select Tidal, Last, or Tidal plus variation (default value).

Output

This dialog allows the user to specify global output options for the simulation. This controls the datasets created by the engine which consist of spatially and temporally varied quantities (values per cell at each output time value).

These output datasets are stored in an XMDF solution file.

Output Times Lists

Due to the potentially long simulation times, CMS-Flow does not output at a constant interval during the simulation. Instead, the user defines one or more lists of times. For each dataset output by the engine, the user selects one time list. The selected quantity will be saved at each time in the time list.

- Lists: CMS-Flow supports up to four different and individual sets of time values.
- Output Times: The times included in the selected time list are displayed in this window.
- Type: The user adds and removes times to/from a time list using the tools in the time type section.

Output Datasets

CMS-Flow can output each of the following datasets. For each group, a time list must be specified. The optional datasets may turned off/on by clicking on their checkboxes in the output dataset tree.

- Water surface elevation
- Velocity
  - Current Magnitude (optional)
  - Current Velocity
- Morphology
  - Depth (through time)
  - Morphology Change (optional)

- Transport
  - Sediment Total-Load Concentration (optional)
  - Sediment Total-Load Capacity (optional)
  - Fraction Suspended (optional)
  - Total Sediment Transport
  - Salinity Concentration (optional)
- Waves
  - Wave Height
  - Wave Period
  - Wave Height Vector
  - Wave Dissipation (optional)
- Wind
  - Wind Speed (optional)
  - Wind Speed Vector
- Eddy Viscosity

Statistical Output

Statistical data can be written out for hydrodynamics, sediment transport, and salinity. This option is turned on by toggles, and a start time, increment, and end time must be specified.

Output Options

Output can be written out in ASCII format and using XMDF file compression. These options are turned on by toggle boxes. All datasets created by the model will be labeled with a simulation label and quantity label. An edit field allows specification of the simulation label.

Advanced Options

This dialog allows the user to specify experimental options when working with the model developers.

Advanced Cards

Model options not available in the interface can be specified in the CMS-Flow model control under the advanced tab. Each row in the spreadsheet will be saved as a different line in the *.cmscards file. The cards can be inserted and deleted using the insert and delete rows.

- The lines will be written out exactly as typed.
- The lines will be written out in the order in which they appear in the dialog.
- Any lines encountered in a *.cmscards file that is not recognized will appear in this list.
- Lines beginning with a "!" character are treated as comment lines and will be skipped over during a read.
- Comment lines are automatically generated and will not be preserved.
- All of these advanced cards will be grouped together at the bottom of the file regardless of where they appeared during reading.

Go to CMS-Flow

References

[1] http://cirp.usace.army.mil/wiki/CMS-Flow_Wind_Forcing#Spatially_Variable_Wind_and_Atmospheric_Pressure

CMS-Flow Observation Cells

CMS-Flow includes observation cells which can be used to output calculations at specific locations. Within SMS, there are two steps to output observation data. First select locations (cells) and specify the cell types as observation stations in the Cell Attributes window. Second, specify what type of data CMS-Flow will output and at what frequency, on the *Cells* tab of *Model Control*.

Model Control

Within the CMS-Flow Model Control window, the type and frequency of observation output to be written can be specified on the *Cells* tab. For each type (time series, flow rate, and transport), individual calculations can be selected and the *Output interval* will be enable if at least one calculation type is selected. *Water surface (ETA)* is calculated at the center of the cell, while all vectors (U and V, X and Y) are calculated at the center of the left face (for horizontal) and the center of the bottom face (for vertical) of the cell.

Display Options

CMS-Flow includes observation symbols to differentiate specifications. On the Cartesian Grid page of the Display Options window (when CMS-Flow is the active model), the *Observation symbols* check box controls the display of symbols that will appear in observation cells. If this is turned on, then the user must be aware of the individual symbol settings accessed by clicking on the *Options...* button. The *Options...* button displays the CMS-Flow Observation Symbols window.

Observation symbols can be selected for the three observation output types (Time series, Flow rate, Transport) and a combination of types. The specific output type symbols are displayed in an observation cell that only includes this type or, depending on the state and choice of *Combination of types* option, also includes this type. Since observation cells can include multiple output types, it is possible to display overlapping symbols. The *Combination of types* combo box specifies the qualification for the combination symbol to be displayed in an observation cell. If the combination symbol is displayed in a cell, then no other observation symbol will also be displayed in that cell. The *Combination of types* can be used to quickly highlight specific observation stations as follows:

- **Any combination** – Any observation cell that includes at least any two output types.
- **Time series and flow rate** – All cells that only include both time series and flow rate output types.
- **Time series and transport** – All cells that only include both time series and transport output types.
- **Flow rate and transport** – All cells that only include both flow rate and transport output types.
- **All types** – All cells that include all three output types.

Observation cells of only one type, such as only time series, can be found by turning on the specific type (*Time series*), turning off the other two types (*Flow rate* and *Transport*), and turning on and setting *Combination of types* to *All types*. The time series only observation cells will be highlighted with time series symbol (ignore the combination symbols).

If the *Observation symbols* check box is turned off, no symbols will be displayed and the individual settings cannot be accessed, however, the individual settings will not be changed.

Model Check

The CMS-Flow Model Checker, accessed from the CMS-Flow | Model Check... menu item, includes a check to ensure that CMS-Flow has been set up to output the specific data for any existing observation station and that observation stations exist for any specified output data. Observation is not required to run CMS-Flow so this is a reminder that something may have been intended but forgotten. It is suggested that the model checker be used prior to running CMS-Flow.

Related Topics

- CMS-Flow

CMS-Flow Output Control

Output files specified here are associated with observation cells that have been assigned within the grid. For example, if a time series observation cell exists, an output file will be written out by M2D for every file type that is checked within this dialog. The same holds true for flow rate observation cells. All observation cell output files are given the file extension of ".m2o". A prefix is specified for all time series and flow rate output files. The user must also specify the time step increment (in seconds) at which to write to the output files. This increment should be a multiple of the simulation time step.

A brief explanation of the information that each of the following observation cell output file types contains is given (* = prefix):

Time Series Output Files:

- U Output (*_u.m2o): Velocity in the x-direction
- V Output (*_v.m2o): Velocity in the y-direction
- ETA Output (*_h.m2o): Water level
- U DETA/DX (u □□/□x) (*_udhdx.m2o):
- V DETA/DY (v □□/□y) (*_vdhdy.m2o):
- ETA DU/DX (□□□u/□x) (*_hdudx.m2o):
- ETA DV/DY (□□□v/□y) (*_hdvdy.m2o):
- X-Momentum Advection U DU/DX (u □u/□x) (*_xmomu.m2o): U component of the momentum advection term in the x-direction
- Y-Momentum Advection U DV/DX (u □v/□x) (*_ymomu.m2o): U component of the momentum advection term in the y-direction
- X-Momentum Advection V DU/DY (v □u/□y) (*_xmomv.m2o): V component of the momentum advection term in the x-direction
- Y-Momentum Advection V DV/DY (v □v/□y) (*_ymomv.m2o): V component of the momentum advection term in the y-direction
- X Bottom Friction (*_xbfric.m2o): X component of the bottom friction
- Y Bottom Friction (*_ybfric.m2o): Y component of the bottom friction
- X Wind Stress (*_xwnd.m2o): X component of the wind stress
- Y Wind Stress (*_ywnd.m2o): Y component of the wind stress

Flow Rate Output Files:

- X Direction (*_qx.m2o): Flow rate in the x-direction

- Y Direction (*_qy.m2o): Flow rate in the y-direction

Return to CMS-Flow Model Control

CMS-Flow Salinity

This tab allows the user to specify model parameters relating to salinity modeling in CMS-Flow. The controls include:

Salinity

CMS-Flow includes the capability to simulate the diffusion of saline water in fresh water. This section includes controls to enable and initialize these calculations.

- Calculate salinity – Turn on the calculation of salinity concentrations in the CMS-Flow simulation.

Time Steps

- In addition to the time step specified in the general parameters tab for hydrodynamic calculations, the explicit model also allows the specification of longer time step for the salinity dispersion process.
- Transport rate – If salinity is enabled in this tab, and the solver option is set to "explicit" in the Flow tab, the user must specify a this time step. It must be larger than the hydrodynamic time step and is generally at least a factor of 20. The default transport rate time step is 60 seconds.

Initial Condition

Salinity transport requires that each cell have an initial value for salinity concentration (measured in ppt). CMS-Flow allows this value to start as a constant for each cell, in which case, during the initial (or ramp time)of the simulation the concentrations are distributing to natural values.

- Global concentration – This control allows the user to specify an initial concentration for the entire domain. The value 0.0 ppt (fresh water) is a commonly used value.
- Spatially varied – these controls allow the user to create or select a spatially varied dataset that will be used by CMS-Flow as the initial concentration at each cell. This dataset is editable by the user.

External Documentation

- Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change[1]

Related Topics

- CMS-Flow

References

[1] http://www.dtic.mil/cgi-bin/GetTRDoc?AD=ADA453954

CMS-Flow Spatial Datasets

Spatially varied data, or spatial datasets are integral to the use and application of the CMS-Flow numerical engine. The engine uses spatial data for input and each process simulated by the engine creates spatial data sets that can then be visualized and used for post processing in the SMS.

Input Datasets

CMS-Flow has several spatially variable input fields. The inputs are sorted by process and labeled with units below.

Hydrodynamics

Variable	Units	Symbol	Description
Bottom Friction	NA	(none)	Roughness of the bottom used to simulate resistance to flow. For example, this could be based on a Manning n value.

Sediment transport/Morphology change

Variable	Units	Symbol	Description
D50	milimeters	(mm)	Median grain size for particles in each cell of the grid domain
Hard Bottom	meters	(m)	Constraint for maximum erosion for each cell of the grid domain

Salinity

Variable	Units	Symbol	Description
Initial Concentration	parts per thousand	(ppt)	Initial salinity concentration for each cell of the grid domain

Output Datasets

Hydrodynamics

Variable	Units	Symbol	Description
Water Surface Elevation	meters	(m)	Elevation above mean sea-level computed at the centroid of each cell.
Current Velocity	meters per second	(m/sec)	Velocity components in the u and v directions computed at the top and right side of each cell. The SMS interface interpolates these values over the cell, converts them to global world (x, y) values and displays these as vectors.
Flow Rate	cubic meters per second	(m^3/sec)	Flux over each cell.

Sediment transport

Variable	Units	Symbol	Description
Concentration Capacity	kilogram per meter cubed	(kg/m^3)	Total transport capacity for flow in the cell. Computed for LUND-CIRP, van Rijn, and Watanabe transport equations.
Sediment Concentration	kilogram per meter cubed	(kg/m^3)	Computed suspended sediment concentration at the centroid of each cell
Sediment Transport Rate	cubic meters per second per meter	($m^3/sec/m$)	Computed transport rate across the right and top faces of each cell. Displayed as a vector quantity in SMS

Morphology change

Variable	Units	Symbol	Description
Bed depth	meters	(m)	Depth below mean sea level to current floor of the model (positive values measured downwards)
Change in depth	meters	(m/sec)	Change in the depth at each cell since the start of the simulation

Salinity

Variable	Units	Symbol	Description
Salinity Concentration	parts per thousand	(ppt)	Concentration at each cell centroid

Related Topics

- CMS-Flow

CMS-Flow Time Control

The CMS-Flow Time Control dialog allows the user to specify the starting day and time for the simulation.

Other options include:

- Start Time – The hour at which to begin the model. This will be the first time step listed in any solution files.
- Simulation Duration – The length of the simulation in hours.
- Ramp Duration – The period of time in days at which to allow the model to "build up" or "ramp up" to a legitimate solution. Aids in the convergence of the model.
- Time Step Size – The time increment at which M2D makes internal calculations. Running the CMS-Flow Model Check from the CMS-Flow menu will provide an initial estimation for the maximum time step size that will produce a convergent result.

Return to CMS-Flow Model Control

CMS-Flow Transport Control

The CMS-Flow Transport Control dialog allows the user to view and edit the sediment and salinity parameters that affect how CMS-Flow runs and what options are to be included in the current simulation. The dialog includes two tabs which partition the parameters into related groups. The tabs and their related parameters include:

Sediment

Sediment Transport – This tab allows the user to specify model parameters relating to both salinity and sediment transport modeling in CMS-Flow. These are two separate options. The tab also includes controls to allow specification of spatially varied data that impact sediment transport. The controls include:

Time Steps

In addition to the time step specified in the general parameters tab for hydrodynamic calculations, the explicit model also allows the specification of longer time steps for processes that do not vary as rapidly. These include the salinity and sediment transport.

- Transport rate – If either salinity or sediment transport are enabled in this tab, the user must specify a this time step". It must be larger than the hydrodynamic time step and is generally at least a factor of 20. The default transport rate time step is 60 seconds.
- Morphologic – If sediment transport is enabled for a simulation the user specifies the morphologic time step. This controls how frequently the bottom elevations of the domain are updated in the hydrodynamic calculations. This time step is generally measured in hours because the change in bed elevations do not occur rapidly enough to impact the hydrodynamics.

Sediment

The Sediment section of this tab turns on the calculation of sediment transport and specifies the formula that CMS-Flow will employ for sediment transport.

- Calculate sediment transport
- Use non-equilibrium transport

Calculate Morphology Change During Ramp Period

This can be toggled on or off.

Avalanching

This can be toggled on or off

Hard Bottom

CMS-Flow includes the option to limit sediment transport on a cell by cell basis to simulate the existence of hard bottom. This section of the dialog allows the user to select, or create a dataset to represent these limiting depths.

- Create Dataset – By default, when SMS creates a "hard bottom" dataset, an inactive value (-999) will be assigned for each cell. This value indicates that no limit to scour exists. The user may select cells and assign limiting depths.
- Select Dataset – If survey data exists that defines the limiting depths of scour, these can be interpolated to the grid, creating a spatially varied dataset. The select button allows the user to select such a set to use as the hard bottom dataset. SMS copies this dataset to an editable set allowing further modification to these values.

D50

CMS-Flow relies on a spatially varied dataset of median particle size to compute the shear stresses used in sediment transport and morphology calculations. This section of the dialog allows the user to create/select the dataset that defines these particle diameters. The buttons operate similar to the buttons for Hard Bottom described above.

- Create Dataset – When creating a dataset, SMS prompts the user for the median particle size that will be assigned as the initial value for all cells.
- Select Dataset

Sediment Standard Deviation

- Create Dataset – When creating a dataset, SMS prompts the user for the median particle size that will be assigned as the initial value for all cells.
- Select Dataset

Salinity

CMS-Flow includes the capability to simulate the diffusion of saline water in fresh water. This section includes controls to enable and initialize these calculations.

- Calculate salinity – Turn on the calculation of salinity concentrations in the CMS-Flow simulation.
- Global concentration – This control allows the user to specify an initial concentration for the entire domain. Generally, a constant value is chosen, with 0.0 ppt (fresh water) being a commonly used value. Since this is not a realistic condition for salinity studies, the first portion of the simulation is often ignored (as with a hydrodynamic ramp) while the model computes spatially varied salinity concentrations. Hot start files can be used to skip this portion of a simulation for subsequent runs.

External Documentation

- Lund Cirp and Watanabe Formula
- Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change[1]

Return to CMS-Flow Model Control

References

[1] http://cirp.wes.army.mil/cirp/pubs/pdf/TR-06-9.pdf

CMS-Flow User-Editable Datasets

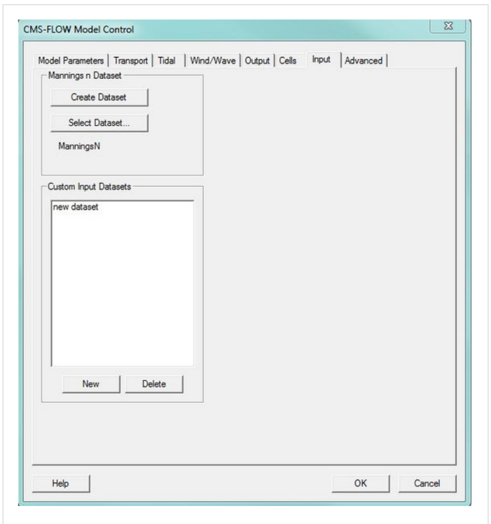
SMS now has the ability to create editable datasets for CMS-Flow simulations. The datasets can be created using a single value or an existing dataset as the initial values.

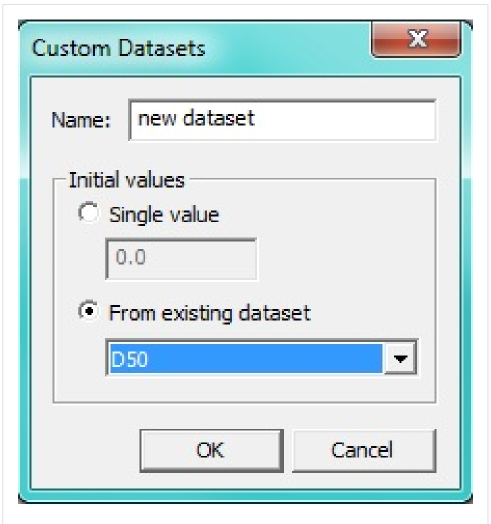
Editable datasets are managed in the Input portion of the CMS-Flow Model Control.

Selecting the New button will bring up the Custom Datasets dialog. Here the user has the options to use a single value or an existing dataset as initial values. The list of datasets in the combo box contain all of the scalar datasets that already exist in the project.

These datasets are saved to an HDF5 file and referenced in the cms_cards file. a new boundary condition, cross-shore, was added. This condition allows water to flow in either direction over the boundary.

Note: This is primarily a developer tool.





Related Topics

- CMS-Flow

Lund Cirp and Watanabe Formula

The Lund Cirp and Watanabe formula can be found on page 16 of the Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR[1].

Transport Slope Coefficient

The Transport Slope Coefficient can be found on page 32 of the Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR[2]. D_s = empirical slope coefficient with typical range of 5 to 30.

The Transport Slope Coefficient can vary site by site and even within a single site domain in that some areas have constraints with naturally occurring steep bed slopes (e.g., channels) and other areas have gentle slopes (e.g. beach profiles, or tidal flats). It is a diffusion coefficent for increasing downhill transport or decreasing uphill transport (if D is >1) This is a good parameter to use as a morphology change calibration factor (along with the scalesus and scalebed coefficients). One thing to note is that what may calibrate well for one area will not calibrate well for another so an average value may be necessary.

Additional Information

Karambas, T.V. 2003. "Nonlinear wave modeling and sediment transport in the
surf and swash zone," Advances in Coastal Modeling, V.C. Lakhan (ed.),
Elsevier Oceanography Series, 67, Amsterdam, The Netherlands, 267-298

Larson, M., Hanson, H., and Kraus, N.C. 2003. "Numerical modeling of beach
topography change," Advances in Coastal Modeling, V.C. Lakhan (ed.),
Elsevier
Oceanography Series, 67, Amsterdam, The Netherlands, 337-365

Chapter 4 of

Horikawa, K. 1988. (ed.) "Nearshore dynamics and coastal processes. Theory,
measurement, and predictive models," University of Tokyo Press, Tokyo,
Japan

Related Topics

- CMS-Flow
- Q&A CMS-Flow

References

[1] Aug. 2006 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR (http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf)

[2] Aug. 2006 - Two-Dimensional Depth-Averaged Circulation Model CMS-M2D: Version 3.0, Report 2, Sediment Transport and Morphology Change TR (http://cirp.usace.army.mil/Downloads/PDF/TR-06-9.pdf)

M2D

M2D support in the SMS interface ended with SMS version 9.2. This was because M2D became the hydrodynamic/sediment transport model in the Coastal Modeling System (CMS). To reflect the involvement in the "system", the name of the model was changed from M2D to CMS-Flow.

The Coastal Modeling System (CMS) is a system of interacting wave, circulation, sediment transport, and bottom-morphology change models centered around a two-dimensional (2D), depth-averaged finite difference circulation model. The CMS was designed for application at coastal inlets where the tide, wind, rivers, and waves may force water and sediment motion. This project-level model is easy to set up and runs quickly as compared to other comparable models. A 3D option is now under testing.

Please see the **CMS-Flow** wiki page for more information about this model within SMS 10.0+

Q&A CMS-Flow

Return to the Main Q&A Page

Q: For a given CMS run, how much longer (by percentage) will that same run be with the sediment transport calculation enabled?

A: The increase in run time will depend on the sediment transport timing parameters that are set by the user and also which sediment transport option is selected. For total load formulations (Watanabe and Lund_CIRP), I generally use a sediment transport time step in the range of 60 to 120 sec and a morphology change time step of 0.25 ht to 1 hr, depending on the application. If the hydrodynamic time step is in the range of say 0.5 to 10 sec (typical for most applications), then the sediment transport calculations are conducted relatively infrequently and add minimal computational burden (you may not even notice the difference). If the advection-diffusion equation is selected for sediment transport, the sediment transport time step is constrained by a Courant-type condition so you have to use a much smaller sediment transport time step that with the total load formulations. You could start out with a sediment transport time step of say 5 times the hydrodynamic time step. If that sediment transport time step is too large, it will be modified internally to maintain stability. The advection-diffusion equation is much more computationally intensive than the total load formulations and takes a lot more time to run. In cases that I have run, using the A-D equation has doubled the computation time. The morphology change time step is not dependent on selection of sediment transport formulation.

Q: According to the user manuel of CMS, when a flow rate boundary condition is given to a cell string the momentum equation is not solved and the flow component normal to the face is prescribed. Is the continuity equation solved?

A: The continuity equation is not being solved since you specify the flow rate. The momentum and continuity equations are used to find the flow rate.

Telescoping Grids

CMS-Flow can use a grid with varying resolution which is referred to as a telescoping grid. Telescoping grids have a base grid with a specified cell size. The cells in the child grid of the base grid have half the cell size (half in both I and J directions). This means each cell is divided into 4 cells.

Telescoping grids are created by creating polygons where you want the different levels of refinement in a CMS-Flow coverage. Each polygon is assigned the maximum allowable size for cells in this area. When you do Map->2D grid, you specify the size of the base grid. SMS will build a telescoping grid with as many levels as required to satisfy the constraints imposed by the polygons.

Note: Telescoping grids in SMS 11.0 are introduced with limited capability and no support. There are known issues such as problems with shading that will not be addressed in SMS 11.0.

5.4.b. CMS-Wave

CMS-Wave

CMS-Wave	
Model Info	
Model type	Designed for accurate and reliable representation of wave processes affecting operation and maintenance of coastal inlet structures in navigation projects as well as in risk and reliability assessment of shipping in inlets and harbors.
Developer	Lihwa Lin, Ph.D. [1]
Web site	http://cirp.usace.army.mil/
Tutorials	General Section • Data Visualization • Observation Models Section • CMS - CMS-Wave (pending)

CMS-Wave (formerly known as WABED) is one of the principal components of the Coastal Modeling System (CMS). WABED stood for "Wave-Action Balance Equation Diffraction" model. The model is a 2-D wave spectral transformation (phase-averaged). The term "phase-averaged" means that it neglects changes in the wave phase in calculating wave and other nearshore processes. This class of wave models represent changes that occur only in the wave energy density. It was originally build to represent theoretically developed approximations for both wave diffraction and reflection in a nearshore domain.

The SMS interface to CMS-Wave includes tools for creating input files as well as post-processing capabilities.

Graphical Interface

The CMS-Wave Graphical Interface contains tools to create and edit a CMS-Wave simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to CMS-Wave. If a grid has already been created for a CMS-Wave simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to CMS-Wave. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the CMS-Wave Menu. See CMS-Wave Graphical Interface for more information.

Model Data Files

Four input files are required for a CMS-Wave simulation. Up to six optional input files may also be utilized depending on the processes being modeled and the selected model parameters. Depending on which options are selected for the simulation, CMS-Wave will generate from one to six output files.

Sample Simulations

Simulations illustrating the model capabilities will be posted here soon.

Model Development History

- July 2006 - Initial CHETN announcing the release of WABED is published. Interface is available through the STWAVE interface in SMS.
- December 2007 – Four new features were recently added to CMS-Wave. These include:
  - wave run-up – The wave run-up calculation includes both wave setup and maximum vertical swash that enter, for example, in beach erosion during storms.
  - wave transmission and overtopping at structures – The calculation of wave transmission and overtopping is possible for either vertical wall or rubble- mound jetties and breakwaters, and for submerged reefs. These wave run-up and wave overtopping structure calculations are necessary, for example, in the study of overwash and flanking of beaches adjacent to jetties.
  - card-format for convenient model control
  - option for running in a Fast Mode – With the new *Fast-Mode* option, CMS-Wave calculates the spectral transformation on five directional bins (each 30-deg angle for a broad-band input spectrum) or on seven directional bins (each 5-deg angle for a narrow-band input spectrum, or on 25- deg angle for wind input) to minimize execution time. In Fast Mode, the wave model is at least five times faster than in the Normal Mode, which operates on 35 directional bins in the half plane. Fast Mode is recommended for users who need rapid calculation in reconnaissance or test applications. Wave information estimated in the Fast Mode is expected to be less accurate than in the Normal Mode because the calculation is based on fewer directional bins. Normal Mode should be specified for final runs.

Related Topics:

- SMS Models Page
- CMS-Flow

External Links:

- August 2008 ERDC/CHL TR-08-13 CMS-Wave: A Nearshore Spectral Wave Processes Model for Coastal Inlets and Navigation Projects [2]
- May 2007 ERDC/CHL CHETN-I-74 WABED Model in the SMS: Part 2. Graphical Interface [10]
- Jul 2006 ERDC/CHL CHETN-III-73 Wave-Action Balance Equation Diffraction (WABED) Model: Tests of Wave Diffraction and Reflection at Inlets [3]
- Jul 2007 ERDC/CHL CHETN-IV-69 Tips for Developing Bathymetry Grids for Coastal Modeling System Applications [8]
- Feb 2006 ERDC/CHL CHETN-IV-67 Frequently-Asked Questions (FAQs) About Coastal Inlets and U.S. Army Corps of Engineers' Coastal Inlets Research Program (CIRP) [18] Updated FAQ Website [2]
- Sep 2008 Modeling of Morphologic Changes Caused by Inlet Management Strategies at Big Sarasota Pass, Florida [3]

References

[1] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Persons;112
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS;583
[3] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-iii-73.pdf

CMS-WAVE Cell Attributes Dialog

The CMS-Wave Cell Attributes Dialog is used to assign a cell type to selected Cells. It is opened using the menu command CMS-Wave | Assign Cell Attributes. The following cell types can be assigned:

Default

The elevation values of these cells will be used in order to determine whether flow exists in each cell or not.

Structure

The cell attributes dialog of CMS-Wave supports various structure types. Each type of structure can specify an attribute value. The following structure types are supported:

- **Bathymetry modification** – for adding alternative feature or structure (immersed or exposed) without modifying the input depth
- **Wave runup** – for calculation of wave runup and overwash on beach face or structure, and adjacent land
- **Floating breakwater** – for calculation of transmitted waves of a floating breakwater
- **Wall breakwater** – for a vertical wall breakwater
- **Rubble-mound** – for a composite or rubble-mound breakwater
- **Piers/docks** – High permeability - for a pier or dock
- **Rubble-mound breakwater** – Low permeability - for a rubble-mound breakwater

Each of the structures can provide an optional attribute value. The value specified is as follows:

CMS-Wave Structure Attributes

Structure Type	Attribute Name	Description	If Not Provided
Bathymetry modification	Depth	Feature structure depth	Assumed land if not provided
Wave runup	Elevation	Beach/structure elevation above mean water level (no effect if < 0.0)	Assumed input depth
Floating breakwater	Draft	Floating breakwater draft (no effect if < 0.05 m)	No effect
Wall breakwater	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Rubble-mound	Elevation	Beach/structure elevation above mean water level (immersed if < 0.0)	Assumed input depth
Piers/docks – High permeability	Porous layer thickness	See permeable structures below.	
Rubble-mound breakwater – Low permeability	Porous layer thickness	See permeable structures below.	

Permeable Structures

CMS-Wave treats cells designated as "Piers/Docks" and "Rubble Mound Breakwater" as if they have a structure that fills the entire water column. If the water level rises, the assumption is made that the structure is still all the way to the water surface (and beyond if needed). CMS-Wave ignores the specified depth for these cells when the structure type is specified. The layer thickness defines the thickness that will convey wave energy. Physically, this is like the distance from the mean sea level down to the footing of the pier group or dock structure. For a rubble mound, it is the distance below mean sea level where the mound sits on solid material (non-permeable).

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells.

Nesting output

Spectral output is generated for nesting output cells cells to be used as input for a nested child grid.

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the STWAVE Display Options.

Related Topics

- STWAVE Menu

CMS-WAVE Graphical Interface

The CMS-Wave graphical interface includes tools to assit with creating, editing, and debugging a CMS-Wave model. The CMS-Wave interface exists in the Cartesian Grid Module.

Model Control

The CMS-Wave Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The CMS-Wave files are written automatically with the SMS project file or can be saved separately using the *File | Save CMS-Wave* or *Save As* menu commands. See CMS-Wave Files for more information on the files used for the CMS-Wave run.

CMS-Wave can be launched from SMS using the *CMS-Wave | Run CMS-Wave* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *CMS-Wave | Model Check* menu command.

Visualizing Results

Select the spectral grid as the grid to use in the Spectral Energy dialog. This will open the spectral grid for viewing in the Spectral Energy dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

CMS-Wave Menu

See CMS-Wave Menu for more information.

Related Topics

- Create a 2D Cartesian grid

CMS-WAVE Menu

The following menu commands are available in the CMS-Wave Menu:

Command	Functionality
Spectral Energy	Opens the spectral energy dialog to define/view wave energy spectra. Generally, CMS-Wave will generate wave conditions internally, but a spectrum may be input. This command also allows the user to visualize wave spectra that are generated inside of the model.
Assign Cell Attributes	Opens the CMS-Wave Cell Attributes Dialog, allowing the user to specify the cell type.
GENESIS Observation Stations	Opens the CMS-Wave Genesis Observation Stations Dialog.
Nest Grid	Opens the CMS-Wave Nesting Options Dialog, allowing the user to specify nesting options.
Merge Cells	Merge multiple rows or columns into a single row or column.
Model Check…	Check for common problems.
Model Control…	Opens the CMS-Wave Model Control dialog to specify model parameters.
Run CMS-Wave	Launches the CMS-Wave model using the currently loaded simulation. As the model runs, a dialog monitors progress of the model and gives the user status messages. When the run is complete, the spatial solutions are read in for analysis and visualization.

CMS-WAVE Model Control

The Model Control… command in the CMS-Wave Menu opens the CMS-Wave Model Control Dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Settings

The settings section is used to specify model input options including:

- Allow wetting and drying
- Forward reflection
- Backward reflection
- Bed friction
- Diffraction intensity
- Wave breaking formula
- Currents
- Muddy bed

Wave Source

The wave source options include:

- **Spectra** – wave state is specified as a wave spectrum on the boundary and the model propagates the spectra over the domain
- **Wind** – waves are generated from wind energy only
- **Spectra and wind** – a combination of the spectra and wind types are used
- **Simplified formulation** – boundaries are specified in the same manner as "Spectra and wind", but the model uses a simplified formulation internally. This can be used for faster run times, however the results will be less accurate.
- **Currents** – specify a dataset and timestep
- **Surge** – specify a dataset or set to be spatially constant
- **Use dates** – associate dates/times with the model
- **Spectra** –
  - **None** – spectral data is not used in the simulation
  - **Single** – the grid's spectral data is used
  - **Spatially varied** – spectral input is used from a spectral coverage
  - **From parent grid** – spectral input is used from a parent grid
- **Wind** –
  - **None** – wind data is not used in the simulation
  - **Constant** – wind data is constant throughout the simulation
  - **Varied** – a vector dataset is used to define the wind data
- **Simplified formulation** – boundaries are specified in the same manner, but the model uses a simplified formulation internally. This can be used for faster run times, however the results will be less accurate.

Output

The output section is used to specify optional output from the model including:

- Radiation stresses
- Sea/swell
- Additional output: sea/swell, sea/swell/total, total

Related Topics

- CMS-Wave

CMS-Wave Model Control Dialog

The Model Control… command in the CMS-Wave Menu opens the CMS-Wave Model Control Dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Settings

The settings section is used to specify model input options including:

- Forward reflection
  - Spatially constant (0.0 – 1.0 range)
  - Spatially varied – select a dataset
- Backward reflection
  - Spatially constant (0.0 – 1.0 range)
  - Spatially varied – select a dataset
- Muddy bed
  - None
  - Spatially varied – select a dataset (0.0 – 0.1 value range, 0.04 recommended)
- Bed friction
- Matrix solver
- Allow wetting and drying
- Nonlinear wave effect
- Runup
- Infragravity wave effect
- Diffraction intensity
- Currents
- Spatial wind field
  - None
  - Spatially varied – select a dataset (values in m/s at 10 m)
- Spectral type
  - Half plane
  - Full plane
  - Full plane with *.spc file
- Dates/Times

Wave Source

The wave source options include:

- **Spectra** – wave state is specified as a wave spectrum on the boundary and the model propagates the spectra over the domain
- **Wind** – waves are generated from wind energy only
- **Spectra and wind** – a combination of the spectra and wind types are used
- **Simplified formulation** – boundaries are specified in the same manner as "Spectra and wind", but the model uses a simplified formulation internally. This can be used for faster run times, however the results will be less accurate.

Output

The output section is used to specify optional output from the model including:

- Radiation stresses
- Sea/Swell
- Total
- Breaking
- ASCII/XMDF format

Related Topics

- CMS-Wave

CMS-WAVE Spectral Coverage

Spatially Varied Boundary Conditions

CMS-Wave has the ability to read in read in spectral data from various locations defined in a nesting file. Within SMS, this spectral data is defined using a spectral coverage. Each node in this coverage can be assigned to one spectral grid, defining the conditions at that location.

Coverage Properties

All spectral grids in a spectral coverage must have the same orientation and size. These grid parameters are defined in the Cover Properties dialog, which can be accessed by right-clicking on the coverage in the project explorer and selecting "Properties...". The following grid attributes can be assigned:

- **Plane type** (full or half) – SMS currently supports only half-plane
- **Angle** – set the grid orientation

Frequency Distribution

- **Number** – Set the number of frequency bands (Number = 30)
- **Delta** – Set the step size (Delta = 0.01) in Hz.
- **Minimum** – Set the minimum frequency (Minimum = 0.04) in Hz.
- **Maximum** – View the maximum frequency (Maximum = 0.33) in Hz.

Angle Distribution

- **Number** – View the number of angle bands (Number = 35).
- **Delta** – Set the step size (Delta = 5) in degrees.
- **Minimum** – View the minimum angle (Minimum = 0.0) in degrees.
- **Maximum** – View the maximum angle (Maximum = 360.0) in degrees.

Creating Spectral Data

To create spectral data at a node in the spectral coverage, right-click on the node and select "Node Attributes...". This will bring up the "Spectral Energy" dialog, from which a spectral grid and spectra can be created. Note that each node may only have one spectral grid at a time. See Generate/Edit Spectra.

Using a Spectral Coverage With a CMS-Wave Model

A CMS-Wave model can be set to use a spectral coverage for spatially varied input by selecting "Spatially varied" in the "Spectra" section of the model control. By clicking on the "Select..." button, you can select which spectral coverage should be associated with the model. See CMS-Wave Model Control.

It is required that number of defined spectral cases in the model control does not exceed the number of datasets associated with spectral input defined in the spectral coverage.

Related Topics

- CMS-Wave

5.4.b.1. CMS-WAVE Files

CMS-WAVE Files

SMS will store the data used for CMS-Wave with the project. When running the model, the files used by the model will be exported. You can find more information about the model specific file formats in the model documentation.

Input Files:

- Required files
  - *projname*.sim
  - *projname*.std
  - *projname*.dep
  - *projname*.eng
- Optional files
  - *projname*.cur
  - *projname*.eta
  - *projname*.struct
  - friction.struct
  - forward.struct
  - reverse.struct
- Auxilliary files
  - *projname*.txt

Output Files

- Spatially varied data (value for each cell)
  - *projname*.wav
  - *projname*.brk
  - *projname*.rad
- Full spectra a selected cells
  - *projname*.obs
  - *projname*.nst
- Additional files
  - selhts.out
  - setup.wave

Related Topics

- CMS-Wave

CMS-WAVE Control File

CMS-Wave parameter files always contain a single line of data defining the parameter options for the simulation. This line will include either 6, 10 or 15 values. SMS writes all 15 values to the input files it creates. The other formats are supported for backward compatibility.

If only the first six values are present, the other nine values will be assigned default values. The default for the diffraction intensity factor is weak diffraction (akap = 1.0). The default for the other eight parameters is 0 (defined in table below).

If ten values exist in the file, bottom reflection is defaulted to off, forward reflection is defaulted to 50% and reverse reflection is defaulted to 30%.

After this single line, additional lines may be present to define the location of observation and nesting cells. CMS-Wave will save additional output (spectra) at these cells into the specified *.obs and *.nst files.

A sample file and table of the parameters is defined below.

File Format

```
iprp icur ibrk irs kout ibnd iwet ibf iark iarkr akap bf arc arkr iwvbk
kout rows (each row includes the I and J indices of a selected output cell
nestout
nestout rows (each row includes the I and J indices of selected nesting output cells
```

Sample File

```
1 0 0 0 2 0 0 0 0 0 4.000000 0.005000 0.500000 0.300000 0
10 15
23 41
4
28 7
27 8
28 9
29 10
```

Parameter definitions

Parameter	Description
iprp	-1, for fast-mode simulation
	0, for wave generation and propagation (uses wind input and spectra)
	1, for propagation only (neglect wind input)
	2, for wind only (implies a zero energy spectra)
icur	0, no current file read
	1, read current file with a dataset for each input spectra
	2, read single data set from current file for all spectra
ibrk	0, no breaking file will be created
	1, breaking file with be created with breaking index for each cell
	2, breaking file with be created with energy dissipation flux saved for each cell
irs	0, no radiation stress gradient file will be created
	1, radiation stress gradient file will be created
	2, radiation stress gradient file and wave setup/maximum water level files will be created
kout	0, no selected output data files will be created
	n, output spectra (.obs) and parameters (selhts.out) files will be created at n specified cells
ibnd	0, simulation will use single spectra (*.eng) file along offshore boundary
	1, simulation will use nesting file (.nst) with linear interpolation of boundary input
	2, simulation will use nesting file (.nst) with morphic interpolation of boundary input
iwet	0, normal wetting/drying (based on water level input)
	1, no wetting/drying (water level input ignored)
ibf	0, no bottom friction
	1, for bottom friction with constant Darcy-Weisbach type coefficient (bf)
	2, for bottom friction with variable Darcy-Weisbach type coefficient (friction.dat)
	3, for bottom friction with constant Manning coefficient (bf)
	4, for bottom friction with variable Manning coefficient (friction.dat)
iark	0, no forward reflection
	1, with forward reflection
iarkr	0, no backward reflection
	1, with backward reflection
akap	diffraction intensity factor (0 for no diffraction, 4 for strong diffraction)
bf	constant bottom friction coefficient
ark	constant forward reflection coefficient (0 for no reflection, 1 for maximum forward reflection)
arkr	constant backward reflection coefficient (0 for no reflection, 1 for maximum backward reflection)
iwvbk	0, for extended Goda wave breaking (Sakai et al. 1989)
	1, for extended Miche wave breaking (Battjes 1972; Mase et al. 2005b)
	2, for Battjes and Janssen wave breaking (1978)
	3, for Chawla and Kirby wave breaking (2002)

Related Topics

- SMS:CMS-Wave_Files
- File Formats

CMS-WAVE Simulation File

CMS-Wave simulation files contain a list of file names that are part of the simulation. This file is created by SMS when the simulation or project is saved. The file begins with a keyword that identifies this as a CMS-Wave simulation file, followed by the world origin and orientation of the grid. The origin is used to display the grid in a global projection (such as UTM or State Plane). It generally corresponds to the location of the corner of the grid offshore. The orientation is the CCW angle from East to the direction of the grid that goes from offshore to on shore. (For example, a coast that opens due West would have an orientation of 0.0, an coast line open to the South would have an orientaion around 270 (or -90) degrees.)

Sample File

```
CMS-Wave 902500.0 4086600 135   /* Key word, grid origin, grid orientation*/
DEP         grid.dep        /* Name of input file containing depth values for each cell */
OPTS        grid.std        /* Name of input file containing model parameters */
CURR        curgrid.cur     /* Name of optional input file containing current values (vx and vy) for each cell */
SPEC        grid.eng        /* Name of input file containing energy density spectra (required in most cases) */
WAVE        grid.wav        /* Name of output file for spatial wave conditions (height, direction) for each cell */
OBSE        grid.obs        /* Name of output file to save full spectra at specified monitoring locations */
NEST        grid.nst        /* Name of output file to save full spectra at specified nesting cells */
BREAK       grid.brk        /* Name of output file to save wave breaking indices or energy disipations at each cell */
SPGEN       grid.txt        /* Name of spectral parameter file, contains parameters used to generate spectra */
RADS        grid.rad        /* Name of output file to save wave radiation stress gradients at each cell */
STRUCT      grid.struct     /* Name of input file containing structure flags for each cell */
SUR         grid.eta        /* Name of optional input file containing spatially varied surge at each cell */
```

The number of particles must be the same for each time step.

Related Topics

- SMS:CMS-Wave_Files
- File Formats

CMS-WAVE Spectral Energy File

CMS-Wave energy files contain a the definition of a half plane spectral grid (5 degree directional bins and user specified frequency bins) and energy density datasets as boundary conditions for each wave condition to be simulated in a CMS-Wave analysis.

File Format

The first line of the file contains the number of frequency bins and directional bins. Typically, 20-30 frequency bins are used. The model requires 5 degree directional bins, so the number of directional bins must be set to 35.

Following the grid dimensions, the next lines of the file specify the frequencies for the model spectra, starting at the lowest frequency. There must be the specified number of values. They are read in free format and may occupy as many lines as needed. These frequencies should span the range where significant wave energy is contained in the spectrum. This can be estimated by inspecting the input spectrum or estimating the peak period expected using the wave growth curves int eh Coastal Engineering Manual. A rule of thumb is that the spectral peak should fall at about the lower one-third of the frequency range (e.g., if the peak frequency is 0.1 Hz, the range may be 0.01 to 0.3 Hz). Wave frequencies higher (or periods shorter) than the highest frequency bin or lower than the lowest frequency bin will not be resolved by the model. Typically, frequency increments are on the order of 0.01 Hz, but the increment need not be constant. West Coast applications will tend to require finer resolution focused at lower frequencies because of long wave periods. Gulf Coast or Gread lakes applications will tend to require coarser resolution covering a broader range of frequencies because of shorter wave periods.

Following the specification of the frequency and direction bins is a header line containing a spectrum identifier, wind information, peak frequency and water elevation correction. This line is read in free format.

After the header line, the file contains the energy densities in the units of meters squared/hertz/radian. The spectrum is read starting with the lowest frequency and reading all the direction s(-85 to 85), then reading energy densities for the next lowest frequency etc.

The file can contain multiple spectra.

Sample File

```
30 35
0.04000 0.05000 0.06000 0.07000 0.08000 0.09000 0.10000 0.11000 0.12000 0.13000
0.14000 0.15000 0.16000 0.17000 0.18000 0.19000 0.20000 0.21000 0.22000 0.23000
0.24000 0.25000 0.26000 0.27000 0.28000 0.29000 0.30000 0.31000 0.32000 0.33000
5 27.000000 0.000000 0.070000 0.000000
0.00000 0.00000 0.00000 0.00000 0.00000 0.00002 0.00006 0.00019 0.00048 0.00107 0.00209 0.00365 0.00576
.
.
.
```

Related Topics

- SMS:CMS-Wave_Files
- File Formats

CMS-WAVE Spectral Table File

Spectral Table Files contain the parameters for generated wave spectra. This file is created by SMS when the Spectral Energy dialog is used to create wave spectra. If wave spectra are imported from buoys or another source, this file will not exist. When this file exists, SMS can use the values in the table to populate the table in the Spectral Energy dialog to allow the user to edit the spectra being used.

SPECTRAL TABLE	Values
Method Option Time Index Angle Hs(1) Tp(1) Gamma (1) Hs (1) Tp(1) Gamma(1) Hs(2) Tp(2) Gamma(2) Wind Fetch nn StdDev Depth	Headers
0 -1 999.0 None 25.0 1.0 20.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	1st row of values
0 -1 999.0 None 30.0 1.0 16.0 8.0 999.0 999.0 999.0 999.0 999.0 30.0 999.0 0.001	2nd row of values

Sample File

ACE/vis drogue path file /* Title */

5 /* Number of Time Steps */

7200.0000 199 /* Current Time Step and Number of Particles */

-0.766993322E+02 0.346589454E+02 1 /* xy values and id */

-0.766986001E+02 0.346616775E+02 2

.

.

8000.0000 199 /* Next Time Step and Number of Particles */

.

.

/* EOF */

The number of particles must be the same for each time step.

Related Topics

- SMS:CMS-Wave_Files
- File Formats

CMS-Wave STD

CMS-Feature enhancements – June 15, 2010 Support the following mode parameters that appear on the first line of the std file (there are a total of 23 of them)

Sample of first line from *.std: 0 0 0 0 0 0 0 0 1 0 4.000000 0.005000 0.500000 0.300000 0 0 1 0 0 0 0 0 0

These parameters represent the following variables in the code:

```
iprp icur ibreak irs kout ibnd iwet ibf iark iarkr akap bf ark arkr iwvbk nonln igrav irunup imud iwnd isolv ixmdf iproc
```

The first 15 parameters should already be supported in the SMS interface and are documented at: xmswiki.com/xms/SMS:CMS-Wave_Control_File [1] The new ones (16th to 23th) and their possible values include:

```
 nonln (nonlinear wave-wave effect): 0 = none (default), 1 = present
 igrav (infragravity wave effect): 0 = none (default), 1 = present
 irunup (run-up): 0 = none (default), 1 = automatic, 2 = field file input (*.runup or runup.dat)
 imud (muddy bed): 0 = field file input (*.mud or mud.dat, default), 1 = none
 iwnd (wind field input): 0 = field file input (*.wind or wind.dat - same format as *.cur, default), 1 = none
 isolv (Matrix solver): 0 = ADI (default), 1 = Gauss-Seidel (allow multiple processors)
 ixmdf (ixmdf output format): 0 = ascii (default), 1 = output in xmdf format, 2 = input/output in xmdf
 iproc (multiple processor): 0 = one-processor (same as 1, default), n = n processors (only for isolv = 1)
 isea (swell and local sea output):
       0 = no additional outputs,
       1 = output additional swell.wav (for swell) and sea.wav (local wave) component files,
          (These have the same format as the total wave field output file *.wav)
       2 = output the total.wav (idential to the default *.wav),
          (This is for the steering run that will merge all individual *swsteer.wav files.)
       3 = output the total.wav, swell.wav and sea.wav.
```

The following new files need to be added in the *.std

```
     *.fric   *.fref   *.bref   *.mud,   *.wind   *.runup
```

(these are the same as friction.dat, forward.dat, backward.dat, mud.dat, wind.dat and runup.dat)

Related Topics

- SMS:CMS-Wave_Files

References

[1] http://xmswiki.com/xms/SMS:CMS-Wave_Control_File

CMS-WAVE Structure File

CMS-Wave structure files contain a line defining the number of cells that should be treated as structures in the model simulation, and then a list of these cells with special features assigned to them. These are specified in the SMS interface by selecting the cell and assigning cell attributes (CMS-Wave menu or right click on selected cell). These cells can represent:

- a floating breakwater
- a bottom mound breakwater
- a beach segment and the land adjacent to it
- jetties or seawalls
- underwater features such as reefs or submerged structures

A trench, submerged mound or structure can be added to the bed as features without modifying the input depth file. Each feature cell is described by four parameters in a line in the structure file. A sample file and table of the parameters is defined below.

File Format

number of structure cells

one row for each structure cell (each row includes the I and J indices of a selected output cell, the structure type and the modifier value

Sample File

4 /* there are 5 structure cells in this sample file */

10 15 1 3 /* cell (10,15) will be modified to have a depth of 3 meters */

23 41 2 0.5 /* model will calculate runup and overwash at cell (23,41). It will be assigned an elevation 0f 0.5 meters above sea level */

15 7 3 1.5 /* model will treat cell (15,7) as a floating breakwater with a draft of 1.5 meters to calculate transmission of waves under the structure */

16 22 4 3 /* model will treat cell (16,22) as a breakwater/seawall/mound with an elevation of 3 meters above sealevel */

Parameter definitions

kstruc	Description	Modifier value (cstruc)
1	add alternative depth without modifying input depth file	specify the new structure depth (assumed land if no depth specified)
2	for calculation of wave runup and overwash on beach face or structure and adjacent land	specify the elevation above mean water level for the cell. Input depth used in no value specified. Has no impact if elevation < 0)
3	for calculation of transmitted wave under a floating breakwater	specify the floating breakwater draft (skipped if not provided or value is < 0.05 m)
4	for a vertical wall breakwater	specify breakwater/structure elevation (input depth value used in not provided; immersed if value < 0.0)
5	for a composite or rubble-mound breakwater	

Related Topics

- SMS:CMS-Wave_Files
- File Formats

5.5. GenCade

GenCade

The GenCade model is a next generation combination of previous long-term planform evolution of a beach models GENESIS (GENEralized Model for SImulating Shoreline Change[1]) and Cascade[2].) and Cascade.

GenCade is a regional model for calculating coastal sediment transport, morphology change, and sand bypassing at inlets and engineered structures. GenCade is developed by the U.S. Army Engineer Research and Development Center (USACE-ERDC), Coastal and Hydraulics Laboratory (CHL). The developers maintain documentation [3] for the model, comments on the interface [4] and steps for a sample application [5].

Functionality

GenCade simulates shoreline change relative to regional morphologic constraints upon which these processes take place. The evolution of multiple interacting coastal projects and morphologic features and pathways, such as those associated with inlets and adjacent beaches may also be simulated. The model is supports responses to imposed wave conditions, coastal structures, and other engineering activity (e.g., beach nourishment).

Typical longshore extents and time periods of modeled projects can be in the ranges of one to 100 km and one month to multiple decades, respectively, and almost arbitrary numbers and combinations of groins, detached breakwaters, seawalls, jetties, and beach fills can be represented. GenCade simulates shoreline change produced by spatial and temporal differences in longshore sand transport. Shoreline movement such as that produced by beach fills and river sediment discharges can also be represented. The main utility of the modeling system lies in simulating the response of the shoreline to structures sited in the nearshore. Shoreline change produced by cross-shore sediment transport as associated with storms and seasonal variations in wave climate cannot be simulated; support of cross-shore processes are being considered for future versions of the model.

Capabilities of GenCade:

- Almost arbitrary numbers and combinations for groins, jetties, detached breakwaters, beach fills, and seawalls
- Compound structures such as T-shaped, Y-shaped, and spur groins
- Bypassing of sand around and transmission through groins and jetties
- Diffraction at detached breakwaters, jetties, and groins
- Coverage of wide spatial extent
- Offshore input waves of arbitrary height, period, and direction
- Multiple wave trains (as from independent wave generation sources)
- Sand transport due to oblique wave incidence and longshore gradient in height
- Wave transmission at detached breakwaters

Related Topics

- 1D Grid Module
- Practical notes for using GenCade
- GenCade modeling process
- GenCade graphical interface

Case Studies / Sample Problems

- Tutorials for learning to use GenCade in SMS are under development.

External Links

- US Army Corps of Engineers Coastal & Hydraulics Laboratory GENESIS website [6]
- GENESIS, Report 1, Technical Reference [7]
- GENESIS, Report 2, Workbook and System User's Manual [8]
- User's Guide to the Shoreline Modeling System [9]
- A History of GENESIS Updates [10]
- Mar 2002 ERDC/CHL CHETN-II-45 Wave Transmission at Detached Breakwaters for Shoreline Response Modeling [37]
- Mar 1990 CETN-II-21 Computer Program: Genesis Version 2 [44]
- Cascade User Guide: [11]
- Cascade Theory and Model Formulation: [12]
- Inlet Reservoir Model (sub-model within Cascade) [13]

References

[1] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=software;34
[2] http://chl.erdc.usace.army.mil/library/publications/chetn/pdf/chetn-xiv-2-pdf
[3] http://cirp.usace.army.mil/wiki/GenCade
[4] http://cirp.usace.army.mil/wiki/GenCade_Users_Guide
[5] http://cirp.usace.army.mil/wiki/GenCade_Example
[6] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=SOFTWARE;34&g=141
[7] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS;110&g=90
[8] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS;111&g=90
[9] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=PUBLICATIONS;112&g=90
[10] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=ARTICLES;457&g=90
[11] https://swwrp.usace.army.mil/_swwrp/swwrp/4-Pubs/TechNotes/swwrp-tn-06-5.pdf
[12] https://swwrp.usace.army.mil/_swwrp/swwrp/4-Pubs/TechNotes/swwrp-tn-06-7.pdf
[13] http://cirp.wes.army.mil/cirp/cetns/chetn-iv39.pdf

GenCade Files

Input

A GenCade simulation requires the following input files:

- *.gen – this is the simulation file and contains links to other input files as well as the definition of all the structures and events and other model parameters.
- *.shi – this contains the initial shoreline position geometry
- *.wave – there must be one wave file for each wave gage. The file contains the wave events for the associated gage for all time spanning the simulation.

The following are optional input files:

- *.shr – this contains the regional contour geometry
- *.shdx – this contains the grid density values for a non-uniform grid

Output

The following files are created by GenCade during a simulation:

- *.prt – this is a summary printed output file that reports on the status of the run. It is for user review.
- *.slo – this file contains the output shoreline positions through the simulation.
- *.off – this file contains the offset from the original shoreline positions. This is the same as the positions fromt he .slo file minus the initial shoreline positions.
- *.qtr – this file contains the transient, spatially varied transport rates at the specified output times during the simulations.
- *.mql/*.mqr/*.mqn – this file contains the annual averaged transport rates to the left, to the right and net. This is a integration of the components that make up the qtr.
- *.irv – One irv file will be created for each inlet in the simulation. It contains the time varying volumes of each of the shoals related to the inlet.

Related Topics

- GenCade

GenCade Modeling Process

A GenCade model is set up in a globally referenced projection (such as UTM or State Plane).

Required Input Data

The model requires the following data for input:

- Grid frame – This is a geometric object that defines the domain of the simulation. The modeling engine operates on a straight line (1D grid). The grid frame defines the origin, orientation and extent of the simulation region. The grid frame should be oriented to align to the principal shoreline direction. Generally the grid frame is positioned just to the landward side of the shoreline. When looking along the shoreline in a direction parallel (or close to parallel) to the direction of the grid frame, the ocean should be but on the left and the shore to the right. Positions in the model (such as shoreline values) are referenced to this datum. The graphical interface will generate a grid that lies along the grid frame. h.
- Initial shoreline definition – This is series of points ((x,y) or (lat,lon)) that define a reference position for the coastline at the beginning of a simulation (in the model projection). SMS stores these points as an arc in a GenCade coverage.
- Wave conditions: The model supports multiple wave inputs at locations throughout the domain. These are mapped to a single cell in the grid. Wave conditions include the wave parameters (height, period, direction) at each time step in the simulation. The direction of each wave condition must be provided to the model in local shore normal coordinates.
- Other structure definitions to represent the structures (seawalls, groins and inlets) in the domain.
- Dredging/nourishment and bypass events as warranted by the events during the period being simulated.

Directions

The model performs all calculations in a local direction defined by the grid. This direction can be visualized relative to several reference points. Two references are defined here:

- The model references the grid direction as an azimuth angle from North. An azimuth of zero (0.0) directs the local grid to go from South to North (land to the East). The azimuth is measured in a clockwise direction, so an azimuth of 90 degrees defines a grid that goes from West to East (land on the South).
- The SMS interface relys more on graphical positioning, since the grid frame is defined interactively and displayed with an origin. However, the system uses a Cartesian angle reference with the X axis (East) equal to 0 degrees. The direction for this convention follows the right hand rule (CCW = positive). Therefore, a Cartesian angle of zero corresponds to the West to East (land on the South) azimuth of 90 degrees. Similarly, a Cartesian angle of 90 degrees is the same as an azimuth of 0 degrees.

The following steps illustrate the standard modeling process.

1. Define a GenCade coverage.
2. Create a grid frame defining the domain extents.
3. Define refinement points as desired to control grid resolution.
4. Define feature points at the location of wave buoys. Read or enter the wave data at each of these points.
5. Define arcs representing the various structures in the domain. Assign attributes to these arcs.
6. Define arcs representing the events in the simulation. Assign attributes to these arcs.
7. Map the conceptual model to a numeric model.
8. In the 1D Grid Module, assign the model parameters in the GenCade Model Control Dialog.
9. Save the simulation and run the model.
10. Read solution files to visualize results.

Related Topics

- GenCade

Practical Notes For Using GenCade

Shoreline change models are not applicable to simulating a randomly fluctuating beach system in which no trend in shoreline position is evident. In particular, GenCade is not applicable to calculating shoreline change in the following situations that involve beach change unrelated to coastal structures, boundary conditions, or spatial differences in wave-induced longshore sand transport:

- Beach change inside inlets or in areas dominated by tidal flow
- Beach change produced by wave-generated currents
- Storm-induced beach erosion in which cross-shore sediment transport processes are dominant
- Scour at structures

Limitations of GenCade:

- No wave reflection from structures
- No direct provision for changing tide level
- Basic limitations of shoreline change modeling theory

Related Topics

- GenCade

5.5.a. GenCade Graphical interface

GenCade Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the GenCade model. The GenCade interface exists in the 1D Grid Module.

A GenCade simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model. To set up a simulation, the modeler can work directly with the 1D grid, but it is much more convenient to work with the conceptual model using a GenCade coverage.

Work with an existing simulation by selecting the 1D Grid Module. If a grid has already been created for a GenCade simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the 1D Grid module active and set the model to GenCade.

The interface consists of the conceptual model tools in the Map Module and the grid specific options in the 1D Grid Module.

Map Module – Conceptual Interface

In the map module, the user creates a GenCade coverage (or coverages) that include the information defining the grid and structures in the desired simulation.

Grid Frame

The grid frame defines the orientation and extent that will be occupied by the 1D grid.

Feature Points

In a GenCade coverage, a feature point can be assigned to be:

- A generic place holder
- A wave gage – The attributes of this point include a time series of wave data.
- A refine point – This point will allow the user to control the grid density in the area of the point.

Feature Arcs

The arcs in a GenCade coverage can be assigned a variety of attributes including:

- Geometric object – shoreline and contours.
- Structure – jetty, inlet, sea wall, or breakwater and the associated parameters.
- Event – bypass or beach nourishment events and their time ranges and parameters.

The conceptual model is converted to a numeric model using the "Map → 1D grid" command. This command can overwrite an existing simulation or simply add additional structures to an existing simulation.

1D Grid Module - Numerical interface

The tools in the grid module allow the user to select individual structures graphically and change their parameters. The GenCade menu allows the user to bring up tables of all structures of a specific type, view their parameters, and make edits as needed.

Gencade Menu

- Global Section
  - Edit Grid
  - Edit Water Level Data
  - Edit Wave Data (New for multiple wave inputs)
  - Filter Wave Data (This tool was part of the GENESIS interface, but is currently not functional for GenCade)
- Shore Parallel Structures Section
  - Breakwaters (Detached breakwaters are temporally invariable at this time)
  - Seawalls
- Shore Normal Structures Section
  - Groins (was combined Groins/Jetties)
- Shore Feature Section
  - Inlets (New – includes an option to specify a jetty at the inlet)
- Events Section
  - Beach Fills
  - Bypassing
- Management Section
  - Model Control
  - Run GenCade

Running the Model

The GenCade Files are written automatically with the SMS project file or can be saved separately using the File | Save GenCade or File | Save As menu commands. See GenCade Files for more information on the files used for the GenCade run. GenCade can be launched from SMS using the GenCade | Run GenCade menu command.

Related Topics

- GenCade Model Control Dialog
- GenCade Result Visualization

GenCade Arc Attributes

The attributes that can be associated with for feature arcs in a GenCade coverage are specified through the GenCade Arc Attributes dialog. Attributes that can be specified for each arc include:

- Arc Type
  - Generic – place holder in the conceptual model. Ignored by the simulation.
  - Initial Shoreline – the geometry of this arc defines the starting shoreline position.
  - Reference Line – this type is not currently used.
  - Regional Contour – the geometry of this arc defines the regional geometric shape.
  - Breakwater – detached structure, generally offshore. In addition to location, the user specifies depths at the ends of the structure and transmission attributes.
  - Seawall – the geometry defined by this arc delineates the furthest landward point the shoreline can erode to in that region.
  - Groin – this arc defines a shore perpendicular structure. The user also specifies the structure permeability and its diffractive attributes.
  - Inlet – this arc actually defines the location of the mouth of the inlet. For each inlet the user defines an inlet name, initial and equilibrium volumes of the shoals associated with that inlet, bypass coefficient and defines dredging events that occur during the simulation. Several other arcs can define other components of the inlet.
  - Left Jetty of Inlet – this arc defines a groin that is part of an inlet complex.
  - Right Jetty of Inlet – this arc defines a groin that is part of an inlet complex.
  - Inlet Attachment Bar – this arc defines the location of the attachment bar that will form related to an inlet complex.
  - Bypass Event – this arc defines the beginning and ending points of a bypass event. The user defines the time ranges and the bypass rates that will occur during the simulation between these points.
  - Beach Fill Event – this arc defines the location of a beach nourishment event. The user defines the time ranges over which material will be added to the beach and the amount of berm width that will be added with each time range.

Related Topics

- Feature Objects Menu

GenCade Events

The GenCade simulation can also include timed events that impact the shoreline. These include dredging events at an inlet, beach fill (nourishment) events, and bypassing events.

Beach Fill

Parameters for characterizing beach fills include begin and end time, location, and width. To define a beach fill, proceed with the following steps:

1. Select the arc associated with the beach fill or select the menu option GenCade | Edit Beach Fill.
2. The GenCade Beach Fill dialog will appear allowing the entry of beach fill parameters. At this point, the user should specify beach fill parameters

Bypassing Event

Parameters for characterizing bypassing events include begin and end time, location, and bypassing rate. To define a bypassing event, proceed with the following steps:

1. Select the arc associated with the bypass or select the menu option GenCade | Edit Bypassing.
2. The GenCade Bypassing dialog will appear allowing the entry of bypassing parameters. At this point, the user should specify parameters

Related Topics

- GenCade

GenCade Menu

The GenCade Menu has the following commands:

- **Edit Grid... –**
- **Edit Water Level... –**
- **Edit Wave Data... –**
- **Filter Wave Data... –**
- **Edit Breakwaters... –**
- **Edit Seawalls... –**
- **Edit Groins –**
- **Edit Inlets –**
- **Edit Beach Fills... –**
- **Edit Bypassing... –**
- **Model Control... –** Can be used to set beach conditions, lateral boundary conditions and general simulation options. See SMS:GenCade Model Control Dialog For more information.
- **Run GenCade... –**

GenCade Model Control Dialog

The Gencade Model Control dialog is used to set beach conditions, lateral boundary conditions and general simulation options. This document highlights the more commonly used options. Refer to the GenCade web site [3] for a more detailed description of how these parameters affect the model results.

Model Setup Tab

The following parameters are specified in the model setup tab:

- Simulation Title
- Simulation start and end time
- Simulation time step
- Recording time step

Beach Setup Tab

The following parameters are specified in the beach setup tab:

- Sand and Beach Data
  - Effective Grain Size
  - Average Berm Height
  - Closure Depth
- Longshore Sand Transport Calibration Coefficients
  - K1
  - K2

Seaward BC Tab

The following parameters are specified in the seaward boundary condition tab:

- Input Wave Adjustments
  - Height Amplification Factor
  - Angle Amplification Factor
  - Angle Offset
- Wave Components to Apply (Primary or Primary & Secondary)
- Number of Cells in Offshore contour Smoothing Window

Lateral BC Tab

The following parameters are specified in the lateral boundary condition tab for the left and right lateral boundary condition:

- Type
  - Pinned
  - Moving
    - Shoreline Displacement Velocity
  - Gated
    - Length of Groin from Shoreline to Seaward Tip

Related Topics

- GenCade

GenCade Result Visualization

Visualization of Results

The interface supports three types of plots to visualize the results of the model. These include:

- Shoreline plots on the grid. These plots are available in the system after reading in the shoreline output (*.slo) file. The offset file (*.off) contains the same data as a displacement quantity and can be read and viewed just like the shoreline output file. The user selects the desired dataset which is then plotted in the graphics window right over the initial shoreline. Display options also allow the display of the minimum and maximum shoreline positions during the simulation. When this file is read, SMS adds several datasets to the project explorer representing:
  - The shoreline position throughout the simulation time. This dataset includes a time step for each output time at each cell in the grid.
  - The change in shoreline position from the initial shoreline (since the beginning of the simulation). This dataset is also transient.
  - The average rate of change of the shoreline during the simulation (single value at each cell of the grid).
- Two dimensional plot of spatially variable datasets. In addition to the *.slo solution file, the user may also instruct SMS to load the sediment transport rate (*.qtr) solution file and the mean transport rate files (*.mqr, *.mql and *.mqn). The sediment transport rate file contains the net transport computed for each cell in the grid at each timestep. The mean transport rates contain annual average transport rates in the right, left, and net directions. This dataset, along with any of the other spatially variable datasets may be visualized in their own plot window. Display options allow the user to select one or more of the datasets to be plotted. The user may also instruct SMS to plot the "active" dataset which results in the plot being updated as the active dataset is changed in the project explorer. In addition, multiple time steps may be selected (or the plot can use the "active" time step). SMS will assign a symbol to each dataset and cycle through line styles for each displayed time step. To create this plot:
  1. Load at least one solution file into the project.
  2. Click on the "Plot Wizard" tool.
  3. Select the "Shoreline" option and click the "Next" button.
  4. Choose the plot options to select the dataset and timesteps to be plotted.
- Two dimensional plot of the Inlet Volume transport. The GenCade model outputs a solution file for each inlet defined in the domain. These solution files (*.irv files), contain sediment volumes and transport rates (in and out) for each of the structures/shoals related to the inlet including the ebb shoal, the flood shoal, and attachment and bypass bars both upstream and downstream of the inlet. The SMS package will search the directory for inlet solution files matching the specified inlet names and populate a list of functions that can be plotted for the selected inlet. Each function will be assigned a unique color to be plotted. The user also selects a time range to plot the volume (or transport rates). To create this plot:
  1. Run a simulation that includes an inlet to generate at least one inlet solution file (*.irv).
  2. Click on the "Plot Wizard" tool.
  3. Select the "Inlet TS" option and click the "Next" button.
  4. Choose the plot options to select the inlet, functions and time range to be plotted.

The user may interact with the plot by dragging a zoom box over an area or right clicking on the plot. Right clicking invokes a menu to select the plot options, frame the data, set the plot attributes, or export the plot data in tabular or image form.

Related Topics

- GenCade

GenCade Structures

GenCade has the capability to model the structures listed below. The GenCade menu includes commands to view a table of the structures of a specific type included in a simulation. Some structures can also be selected interactively if they have a geographic attribute such as length.

Breakwaters

Parameters for characterizing breakwaters include start and end location, depth, and transmission coefficient. To define a breakwater, proceed with the following steps:

1. Select the *Create Breakwater* tool and click with the left mouse button to specify the start and end location.
2. Select the menu option *GenCade* | *Edit Breakwaters*. The *GenCade Detached Breakwaters* dialog will appear allowing the entry of breakwater parameters. At this point, the user should verify that the parameters are correct. Further information concerning the selection of breakwater parameters is provided in the GenCade model documentation.

Seawalls

Parameters for characterizing seawalls include start and end location. To define a seawall, proceed with the following steps:

1. Select the *Create Seawall* tool and click with the left mouse button to specify the start and end location.
2. Select the menu option *GenCade* | *Edit Seawalls*. The *GenCade Seawalls* dialog will appear allowing the entry of seawall parameters. At this point, the user should verify that the parameters are correct.

Groins

Parameters for characterizing groins include length, permeability, diffraction, and depth. To define a groin or jetty, proceed with the following steps:

1. Select the *Create Jetty/Groin* tool and click to specify the jetty or groin location. The groin or jetty will be created perpendicular to the grid.
2. Select the menu option *GenCade* | *Edit Groins & Jetties*. The *GenCade Groins & Jetties* dialog will appear allowing the entry of groin and jetty parameters. At this point, the user should verify that the parameters are correct. Further information concerning the selection of groin and jetty parameters is provided in the GenCade model documentation.

Inlets

To create an inlet, create a feature arc, right click on it, select *Attributes* and change the attribute to *Inlet*. Inlets have a name, a left and right bypass coefficient, cell positions, shoal volumes, jetties, and dredging options.

Cell Positions

Cell beginning and ending positions can specified for the inlet and its left and right bypass. These can be set by creating attachment arcs by the inlet before mapping to a 1D grid. If there are two inlets with one attachment arc in between, the attachment arc will be associated to the inlet closest to the arc.

Shoal Volumes

The initial and equilibrium shoal volumes can be specified for the ebb, flood, bypass (left and right) and attachment (left and right) shoals.

Jetties

The jetties to the left and right of the inlet can be given a length, permeability, and a seaward depth. Also, whether the jetty exists and is diffracting can also be specified.

Dredging Events

Dredging events can have a beginning and ending date, which shoal is to be mined, and the amount to be removed from the specified shoal between the beginning and ending date.

Related Topics

- GenCade

Wave Gages

Wave gages provide boundary condition information to a GenCade simulation. The wave height, period and direction influence the transport of material on the shoreline and the shoreline morphology. The GenCade model allows the user to define multiple wave gages, providing for spatially varied driving force.

The wave gage is created either as an "Attribute" on a feature point in the GenCade coverage, which will be mapped to a specific cell in the grid, or it may be specified directly on the grid. For each wave gage the user also specifies a depth and wave events. Each wave event has a date and time, a H0, a period, and a direction. The model computes the wave conditions for each cell in the grid by interpolating the data from the neighboring wave gages.

Wave gages specified in a GenCade coverage will be displayed at the feature point in the coverage, and will appear when the 1D grid (not grid frame) is created on the associated cell in the grid.

If the user changes attributes of a wave gage on a feature point, these changes do not impact the simulation unless teh attributes are mapped to the grid again (Map → 1D grid). Wave gages can be edited using the "Edit Wave Data.." option from the GenCade menu after the grid is created and still take effect.

Wave Events

Wave events can be edited by clicking on the "Data..." button when editing the wave gage. This can be done for either a gage on a 1D grid, or a gage at a feature point.

The direction of a wave, for a given wave event, for a given wave gage may be specified in a meteorological, oceanographic, or Cartesian convention. Also, the wave direction can be specified in a shore normal convention as long as a 1D grid or 1D grid frame is present in the coverage which contains the wave gage. The default convention for wave gages is meteorological.

Wave Data for GenCade

Wave conditions, including significant wave height, peak period, and direction, drive the transport forces that impact coastal morphology. GenCade requires wave data for at least one location in the domain, and supports options for the user to specify wave data at multiple locations in the domain. Each wave gage includes:

- A location – (X,Y) coordinate as a feature point and Cell ID as an input to the model.
- A depth – the water depth at the gage.
- A time series of wave data events – The user obtains these events either from a buoy or from a local wave transformation model.

Issuing the command to edit wave data invokes the *Wave Gages* dialog. This dialog lists the gauge cell IDs and the depths. It also includes a button for each gage to view/edit the wave data at that gage.

Clicking on the *Data ...* button for any of the gages brings up the *Wave Events* dialog. This dialog displays the time, height, period and direction (in shore normal coordinates) for the gage. The data is in a spread sheet, so it may be copied and pasted from other spread sheet data. There is also an "Import" button that can be used bring up the import wizard to bring in wave events data stored in a text file. The data is saved and passed to the model in the wave file.

Related Topics

- GenCade

5.6 STWAVE

STWAVE

STWAVE	
Model Info	
Model type	Model for nearshore wind-wave growth and propagation.
Developer	Jane Smith [1]
Web site	STWAVE web site [2]
Tutorials	General Section • Data Visualization • Observation Models Section • STWAVE Several sets of sample problems and case studies are available. These include: • Model Validataion cases from the STWAVE website [2]

STWAVE is a steady-state, finite difference, spectral model based on the wave action balance equation. STWAVE is written by the U.S. Army Corps of Engineers Waterways Experiment Station (USACE-WES).

Functionality

STWAVE simulates depth-induced wave refraction and shoaling, current-induced refraction and shoaling, depth- and steepness-induced wave breaking, diffraction, wave growth because of wind input, and wave-wave interaction and white capping that redistribute and dissipate energy in a growing wave field. The purpose of STWAVE is to provide an easy-to-apply, flexible, and robust model for nearshore wind-wave growth and propagation. Recent upgrades to the model include wind, surge and friction fields (spatially varied). Also, wind and surge fields can be temporally varied. The method of analysis used by the STWAVE code along with the file formats and input parameters are described in the STWAVE documentation. SMS supports both pre- and post-processing for STWAVE.

The new full-plane version of STWAVE is not a replacement for the half-plane version, but a supplement. The half-plane version will always have an advantage of substantially lower memory requirements (~ two orders of magnitude) and faster execution. The half-plane limitation is generally appropriate for nearshore coastal applications, with the exception of enclosed or semi-enclosed bays, estuaries, and lakes where seas and swells may oppose each other or there is no clear “offshore” direction. The full-plane version allows wave input on all boundaries and wave generation from all directions.

Using the Model / Practical Notes

- A grid for use with STWAVE is created and edited in SMS using the Map Module.
- The modeling parameters required by STWAVE are generated and applied to the mesh using commands grouped in the STWAVE menu.
- Post processing of solution data generated by STWAVE is done using the generic visualization tools of SMS.
- Wind can be entered in the STWAVE model control as either a constant value or by specifying an existing Cartesian Grid data set.
- STWAVE requires metric units. All data in SMS needs to be in metric units before running STWAVE.
- Water depths are defined as positive numbers and land elevations are negative numbers.

Graphical Interface

The STWAVE Graphical Interface contains tools to create and edit a STWAVE simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to STWAVE. If a grid has already been created for a STWAVE simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian grid module active and set the model to STWAVE. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian grid menus and tools augmented by the STWAVE Menu. See STWAVE Graphical Interface for more information.

External Links

- Aug 2007 ERDC/CHL CHETN-I-76 Modeling Nearshore Waves for Hurricane Katrina [6]
- Aug 2007 ERDC/CHL CHETN-I-75 Full-Plane STWAVE with Bottom Friction: II. Model Overview [7]
- Sep 2006 9th International Workshop On Wave Hindcasting and Forecasting Jane McKee Smith Modeling Nearshore Waves For Hurricane Katrina [12]
- Mar 2006 ERDC/CHL CHETN-I-71 Full Plane STWAVE: SMS Graphical Interface [17]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [32]
- Jun 2002 ERDC/CHL CHETN-I-66 Grid Nesting with STWAVE [34]
- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [35]
  - Please see this forum post [2] for an explanation of ADCIRC and STWAVE steering
- Sep 2001 ERDC/CHL CHETN-I-64 Modeling Nearshor Wave Transformation with STWAVE [34]

References

[1] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Persons;170
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=SOFTWARE;9

Grid Nesting

Grid nesting refers to the ability to take output at specified locations from one grid to use for boundary conditions on another. We refer to the source grid as the parent grid and the grid using the boundary conditions as the child grid. This approach is often used in order to cover a large area with a coarse grid with a more refined grid in a specific area of interest.

Within SMS you can setup nesting for STWAVE using a STWAVE parent grid or a WAM parent grid. The functionalities are quite similar with some minor changes specific to each type of setup.

Nesting STWAVE → STWAVE

You can do nesting with full or half plane grids but the parent and child grids must be consistent (you can't nest a half-plane STWAVE grid within a full-plane grid). You can specify the nesting options by clicking on the options button in the model control dialog.

Options

- **Select Nesting Grid –** This command indicates that the current grid should obtain input spectral values from a parent grid. By selecting this option the current grid is nested in the grid selected in the combo box below.
- **Type of Boundary Interpolation –** Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

When doing STWAVE → STWAVE nesting, some of the parameters in the child grid must match those in the parent grid. These include whether or not the model is using time steps, whether the model is using a reference time, whether the model is full or half plane, and the identifiers for each of STWAVE snaps (time steps or cases). These options will be disabled in SMS once you have chosen to do nesting.

Nesting WAM → STWAVE

Even though the output spectra from WAM is always full-plane, a full or half plane STWAVE grid may be nested within a WAM grid. You can set the nesting options by clicking on the options button in the model control dialog. Since WAM models are always run at specific date/times, child STWAVE models always use time steps with reference times.

Options

- **WAM Simulation –** The WAM simulation which will provide the input spectra.
- **WAM Grid –** The WAM grid within the simulation to extract spectra.
- **Start time –** The first time to run STWAVE. This time must be an output time specified in the WAM grid options.
- **End time –** The last time to run STWAVE. This time must be an output time specified in the WAM grid options.
- **Interval –** The snap interval must be set so each snap will hit a WAM output time. For example if WAM data was being output every hour, the interval may be 1 hour, 2 hours, or 3 hours but not 1.5 hours.
- **Type of Boundary Interpolation –** Choose to use either linear boundary interpolation or morphic boundary interpolation. The morphic interpolation method was developed to preserve the shape of the directional distribution. It is appropriate for climatic wave transformation studies where a parametric spectral shape is applied based on wave parameters.

The start time, end time, and interval will determine the number of snaps used by STWAVE and the times associated with each.

Spectral Sites

Regardless of the type of nesting that is going to take place, spectra need to be provided to STWAVE along the boundaries with input spectra (see Boundary Conditions Dialog). Both WAM and STWAVE support the ability to output computed spectra at specified locations. These locations can be specified manually using cell attributes but preferably are automatically generated. To automatically generate these output locations, click on the spectral sites button in the model control dialog (the nesting options should have already be set). Be sure that if you are using a full-plane model, the sides which will have spectra input have already been identified. The dialog that comes up will allow you to choose between different options to control the number of nesting cells along the offshore boundary. The more nesting cells you have the better you can capture differences along the boundary. However, more nesting cells require more disk space and time to read and write the files.

Q&A STWAVE

Return to the Main Q&A Page

Q: There appears to be no way of outputting observation wave parameters results to a text file *.out as is the case with the non SMS version of STWAVE. Has this feature been removed?

A: The .obs file is outputted as an h5 file, so you would need an h5 viewer to view the files. They did this to make the file sizes smaller and run faster.

Q: The observation file is called *_obs.h5 and not *.obs as should be the case. Therefore I cannot import the observed wave spectra. Is this a program bug, or is there another way of looking at the observation point data?

A: You can import the obs.h5 file into SMS and view it, you just have to select which one you want to view in the spectral energy window.

Q: Is the output wave direction relative to the grid regardless of specifying global system for boundary conditions?

A: The output is relative to the global system

Q: How does the 1D transformed spectrum work along boundaries adjacent to main wave input boundary?

A: The energy found from STWAVE running in one direction is used when STWAVE runs in the other directions, it can have up to four boundary conditions in full plane.

Q: What is the largest grid domain that the model can handle? (I appear to be crashing the model for no apparent reason but I have quite a few grid cells)

A: It all depends on how much ram you have. If you are running full plane then it will be alot smaller, since it requires alot more to do all the calculations.

Q: Can you nest grids yet in STWAVE?

A: You can if it is in half plane.

Q: How do you get the nest grids option ungreyed out?

A: You have to make sure both grids are half plane, when you create a new one it will be full plane. Then make sure the new created grid has the same dx, dy values in the cell size section.

Saving STWAVE

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- stw_grds.h5 referenced to new save location
- spec_grds.h5 referenced to new save location
- *.grd referenced to new save location
- *.ctl referenced to new save location
- *.sol referenced to a folder in the new save location

STWAVE Boundary Conditions

The external boundary condition for STWAVE consists of one or more energy spectra entering on one or more open edges of the grid. The boundary conditions dialog specifies the STWAVE boundary conditions and locations of these boundary conditions.

Grid Display

This graphic shows the orientation of the grid and labels the sides of the grids which is used in other controls within the dialog.

Edge Boundary Type

The type of boundary condition applied to each edge of the STWAVE grid is shown and in some cases edited in this section of the dialog.

The types of boundary conditions include:

- Specified spectrum – This may come from a parent grid if using nesting otherwise specified below.
- 1D transformed spectrum – This boundary type allows energy to propagate along the boundary without interference. The cells would have the same energy if the grid was extended and the boundary became interior to a larger grid.
- Zero spectrum – The boundary doesn't have any spectral energy applied.

For half-plane models, the boundary condition types are fixed and energy travels in the positive I direction of the grid. In this case, the user specifies spectra that will be introduced at side 1. Sides 2 and 4 will be treated as 1D transformed spectra.

For full-plane models, the boundary condition types may be specified. You cannot have specified spectra on two adjacent boundaries. Therefore, you can have specified spectra on a maximum of two boundaries and these must be on opposite boundaries.

Events Spreadsheet

This spreadsheet defines the time step or cases that will be used and the input boundary conditions for each. SMS is setup so that you can reuse the same spectral information for multiple time steps or cases.

The first column is the case name or the time step value depending upon whether or not the simulation is using time steps. If time steps are not used, the id for the cases can be any alpha-numeric text so long as it does not contain a comma(,) or slash(/)anywhere in the text. However, if time steps are used, then the case id must be an integer number. When time steps are used, the number represents how much later the time is. Hence, if 5 is entered for the case id, and the time units is hours, then it is the case of 5 hours later. When nesting is used, the case ids, and the number of cases, will be determined by the parent simulation. When the parent simulation is a WAM run, then time steps with a reference time must be used.

In addition to the external condition, the engine can simulate distributed forces over the domain including wind, surge and currents. The currents applied to a simulation are specified in the model control dialog. Wind and surge values are specified in the spread sheet in columns 2 through 4.

If you are not using nesting, there will be one or two columns on the end to represent input spectra (two columns will only show up if two boundaries have specified spectrum boundary types). These columns are used to specify which of the spectra to use for each case or time step. The spectra must be generated previously using the Spectral Energy Dialog. You can specify the spectra to use for a time-step or case by selecting the cell in the spreadsheet and then double clicking on the appropriate spectra entry in the energy spectra tree.

The number and use of the remaining columns in the spreadsheet will depend upon the options used for the STWAVE simulation. For example, if a constant value is used for wind and/or surge columns will appear that represent the wind direction, magnitude and/or tidal elevation as applicable.

Energy Spectra

The energy spectra group box contains controls to specify which spectra data to use for each case/time step as described above. The tree represents the active spectral grid and its associated dataset. You can go the spectral energy dialog by clicking on the spectral energy button. Here you can change the active spectral grid and/or create new spectral grids and datasets.

Angle Convention

The angle convention controls allow you to choose the convention that will be used for the wind direction field in the events spreadsheet. The direction represented by the wind angle of the active row of the spreadsheet is plotted on the direction graph.

STWAVE Cell Attributes

The STWAVE Cell Attributes Dialog is used to assign a cell type to selected Cells. It is opened using the menu command STWAVE | Assign Cell Attributes. The following cell types can be assigned:

Default

Cells are no longer assigned as Land or Ocean types but are assigned as Default where their elevation values are used to simulate whether flow exists or not. If no elevation data is available for part of the grid area, it is important to specify an extrapolation value of a negative number. Extrapolation values can be specified in the Interpolation dialog while doing Map → 2D Grid.

Monitoring Station

Spectral output is generated for cells designated as monitoring station cells.

Nesting output

Spectral output is generated for nesting output cells cells to be used as input for a nested child grid.

Displaying Cell Attributes

The symbols and colors used to identify the attributes assigned to cells can be changed in the STWAVE Display Options.

Related Topics

- STWAVE Menu

STWAVE Graphical Interface

The STWAVE graphical interface includes tools to assit with creating, editing, and debugging a STWAVE model. The STWAVE interface exists in the Cartesian Grid Module.

Model Control

The STWAVE Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls, run types, output options, global parameters, print options and other global settings.

Running the Model

The STWAVE files are written automatically with the SMS project file or can be saved separately using the *File | Save STWAVE* or *File | Save As* menu commands. See STWAVE Files for more information on the files used for the STWAVE run.

STWAVE can be launched from SMS using the *STWAVE | Run STWAVE* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *STWAVE | Model Check* menu command.

STWAVE Menu

See STWAVE Menu for more information.

Visualizing Results

The solution files for STWAVE are XMDF files. These files can be opened in the normal SMS interface. To view the observation or nesting spectral output simply open the files (outside of the Spectral Energy dialog), open the Spectral Energy dialog, and select the spectral grid. This will open the spectral grid for viewing in the Spectral Energy dialog. Make sure to set the original grid back before leaving the dialog to ensure the model does not change.

Related Topics

- Create a 2D Cartesian grid

STWAVE Menu

The items in the STWAVE menu in the Cartesian Grid Module are described below:

- Spectral Energy
- Assign Cell Attributes
- Model Control
- Run STWAVE

Related Topics

- STWAVE

STWAVE Model Control

The Model Control… command in the STWAVE Menu opens the STWAVE Model Control Dialog. This dialog is divided into sections for different types of parameters which are used by the model as it runs. These include:

Grid Definition

The grid definition section of the model control reports defining characteristics of the grid including:

- Cell size
- Number of rows and columns
- X, Y origin
- Angle of rotation

Boundary Condition Source

The boundary condition for STWAVE consists of one or more energy spectra entering on one or more open edges of the grid. Traditionally, the user specifies a single spectra that is assigned to all of the cells on the offshore edge. In this situation the user specifies one spectra for each wave case or time snap being simulated. Alternately, the input spectra can be interpolated to the cells on the offshore edge(s) from a parent STWAVE or WAM grid.

- **Source combo box –** This combo box lists the types of spectral sources. The user chooses from "Specified", "STWAVE run", and "WAM run". If the option is "Specified", the user must provide the spectra by defining or reading them in using the Spectral Energy dialog. Both of the other options indicating that spectra will be coming from a parent grid and the associated options and spectral sites buttons become enabled. If an option is selected that is not valid (i.e. choosing an "STWAVE run" when only a single grid exists in the project so not parent grid exists), SMS generates a message indicating the option is invalid and resets the source.
- **Options... –** This button brings up a dialog which allows the user to specify information for using output from a STWAVE or WAM parent grid as spectral input for this grid. When a parent STWAVE grid is desired, the user specifies which grid is the parent and what type of spectral interpolation should be used.
- **Spectral sites ... –** This button invokes a dialog that creates nesting sites in the parent grid. Spectral output sites for nesting may also be specified explicitly by selecting cells in the parent grid and assigning attributes for those cells. See grid nesting for more information.

Settings

- **Set Boundary Conditions –** Opens the STWAVE Boundary Conditions dialog used to specify the STWAVE Boundary Conditions.
- **Output Control –** Opens the STWAVE Output Control dialog.
- **Full Plane –** Consult the STWAVE model documentation for more information.
- **Half Plane –** Consult the STWAVE model documentation for more information.
- **Source Terms –** Specifies whether STWAVE should generate waves using wind and the input spectrum or whether we are using the wave spectrum.
- **Current Interaction –** Specifies whether currents will be used as input to the model.
- **Bottom Friction –** Specifies which friction type is to be used, if any.
- **Surge Fields –** Specifies whether surge fields are being used, or a constant tidal offset.
- **Wind Fields –** Specifies whether wind fields (spatially varied wind) are being used, or a constant wind magnitude and direction.
- **Use Time Steps –** STWAVE can output solutions as cases or time steps. In order to output as time steps, the "Use Time Steps" toggle must be checked. If using time steps the output units and a reference time (optional) can be specified.
- **Number of Iterations –** Number of iterations to run. (Full Plane Only).
- **Select buttons –** The select buttons by next to the current, bottom friction, surge, and wind options are used to specify a data set on the grid to use as the desired data set. These buttons open up the Select Data set dialog. These data sets must be created prior to entering the model control dialog.

Using Datasets as Input

Datasets may be used as input to STWAVE to represent spatially varied currents, bottom friction (either manning's or JONSWAP), surge, or wind. Bottom friction cannot vary from snap to snap. The other dataset types can have different values for each snap. How datasets are chosen and used depends on whether or not the STWAVE simulation is using time steps (cases or time steps).

If the simulation is using time steps, the datasets are chosen within the model control tab by clicking on the choose button to the right of the dataset choice (it will appear when the dataset option is chosen if STWAVE is using times). The dataset values for each timestep will be the values for the time that the time step occurs. The dataset chosen does not need to have the same number of time steps as the simulation. The dataset values passed to STWAVE can be interpolated from a dataset with values at different time steps than is being used by the model.

If the simulation is not using time steps, the datasets are chosen in the STWAVE Boundary control dialog. In this situation, a steady state dataset is chosen for each snap.

Related Topics

STWAVE Menu

STWAVE Output Control Dialog

The STWAVE Output Control Dialog is used to specify the solutions files STWAVE should output.

The following datasets are always written:

- **Height**
- **Period**
- **Direction**
- **Wave Vector**(magnitude of height with wave direction)
- **1/fma –** Period that represents the inverse of the spectral peak frequency redefined for local wind growth cases

The options the user can set include:

- Radiation stresses – If this is on, radiation stresses will be calculated and output into datasets
- Breaking – This will output a dataset representing the wave breaking. For full-plane, this can be no indices (off) or write indices which gives a value of 1 where breaking occurs and 0.0 otherwise. Half-plane has the full-plane options and has an additional option to calculate energy dissipation. This will give a dataset of energy dissipation.
- Output type – SMS supports the ability to write ASCII files, XMDF files or both. ASCII files refer to STWAVE version 6 global dataset files and XMDF files are datasets written using the XMDF library built upon HDF5. XMDF is the recommended file format because they are much faster to read/write. XMDF is only available on windows platforms.

Related Topics

- STWAVE Model Control

5.7. WAM

WAM

The global ocean WAve prediction Model called WAM is a third generation wave model. WAM predicts directional spectra as well as wave properties such as significant wave height, mean wave direction and frequency, swell wave height and mean direction, and wind stress fields corrected by including the wave induced stress and the drag coefficient at each grid point at chosen output times.

The model is continually updated to incorporate the latest results of research. The verification has been carried out in three areas National Oceanic and Atmospheric Administration (NOAA) moored buoys are available on the global Telecommunications System (GTS). It is hoped that the buoys chosen will allow the identification of both successes and failures in WAM model physics and will minimize shortcomings due to sub-grid scale effect.

Features

- The present version of WAM makes the following assumptions:
- Time dependent wave action balance equation.
- Wave growth based on sea surface roughness and wind characteristics.
- Nonlinear wave and wave interaction by Discrete Interaction Approximation (DIA).
- Free form of spectral shape.
- High dissipation rate to short waves.

Graphical Interface

The WAM Graphical Interface contains tools to create and edit a WAM simulation. The simulation consists of a geometric definition of the model domain (the grid) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the Cartesian Grid Module and setting the current model to WAM. If a grid has already been created for a WAM simulation or an existing simulation read, the grid object will exist in the Project Explorer and selecting that object will make the Cartesian Grid Module active and set the model to WAM. See Creating 2D Cartesian Grids for more information.

The interface consists of the Cartesian Grid Module menus and tools augmented by the WAM Menu. See WAM Graphical Interface for more information.

External Links

- Influence of Marsh Restoration and Degradation on Storm Surge and Waves [1]
- Dynamics and modelling of ocean waves By G. J. Komen, L. Cavaleri, M. Donelan, S. Hasselmann, P. A. E. M. Janssen

References

[1] http://www.mvd.usace.army.mil/lcast/pdfs/chetn-i-77.pdf

WAM – Map to Raster Utility

Map to Raster

- The Map to Raster utility has been updated to allow for conversions between big endian to little endian. Most machines are little endian and therefore users should just use the utility like they did before. However, if the InOut/Map files were created from a machine that is big endian the user should convert to little endian. To convert: copy the InOut/Map files to a little endian machine. Then from the command line run map_to_rast.exe passing "big_endian" as an argument. It will look something like this: c:\>map_to_rast.exe big_endian. Other command line arguments can also be used outlined below:

native

```
Specifies that unformatted data should not be converted.
```

big_endian

```
Specifies that the format will be big endian for integer data and big endian IEEE floating-point for real and complex data.
```

cray

```
Specifies that the format will be big endian for integer data and CRAY* floating-point for real and complex data.
```

fdx (Linux, Mac OS X)

```
Specifies that the format will be little endian for integer data, and
VAX processor floating-point format F_floating, D_floating, and
X_floating for real and complex data.
```

fgx (Linux, Mac OS X)

```
Specifies that the format will be little endian for integer data, and
VAX processor floating-point format F_floating, G_floating, and
X_floating for real and complex data.
```

ibm

```
Specifies that the format will be big endian for integer data and IBM* System\370 floating-point format for real and complex data.
```

little_endian

```
Specifies that the format will be little endian for integer data and little endian IEEE floating-point for real and complex data.
```

vaxd

```
Specifies that the format will be little endian for integer data, and
VAX* processor floating-point format F_floating, D_floating, and
H_floating for real and complex data.
```

vaxg

```
Specifies that the format will be little endian for integer data, and
VAX processor floating-point format F_floating, G_floating, and
H_floating for real and complex data.
```

5.7.a. WAM Graphical Interface

WAM Graphical Interface

Overview

The WAM interface supports the wave model WAM.

WAM models are built by creating WAM grids, WAM simulations (right-click in blank area of project explorer), and dragging the WAM grids you want to use into each WAM simulation. There are options specific to each grid in a simulation as well as the simulation as a whole (model control).

WAM has the ability to use nested grids to improve resolution in an area of interest.

Grid Options

Once the WAM Grid has been created, different options have to be provided. In the Grid options dialog, there are three tabs.

- The General Tab
- The Output Tab
- The Spatial Input Tab

WAM Simulation

A new WAM simulation is created by right-clicking in the Project Explorer and selecting WAM from the New Simulation menu. The WAM grid is then dragged under the new simulation to create a link. If there are multiple grids in the same project (for example, nested grid and parent grid), all grids should be linked to the simulation.

Multiple simulations can also be created.

Model Control

Once a simulation is created and the grid(s) is linked, the grid options can be edited in the Simulation Model Control which can be accessed by right-clicking on the simulation. Once all information are entered, WAM files can be exported, the project can be saved and WAM can be launched.

Model Wrapper

Once all of the simulation information is ready, the WAM files can be exported by right clicking on the simulation. After the WAM files are exported, then the model wrapper can be launched by right-clicking on the simulation and choosing "Launch WAM". Both steps can be done with the "Save Project, Export, and Launch WAM" option, also under the right-click menu.

SMS will export all WAM files to a folder with the same name as the simulation under the WAM folder in the same directory as the saved SMS project. All wind dataset files will be in a folder called "WindInput" under the simulation folder. All WAM files, except the wind dataset, for each grid in the simulation will be in a folder with the same name as the grid under the simulation folder.

After all the files for a WAM simulation have been exported, the WAM executables can be launched. There are three WAM executables which are, preproc, chief and map_to_rast. These must be executed in order for a given grid.

Preproc creates the Grid_Info and Preproc_prot files. Chief is the main process of WAM, and it creates the boundary value output files, the integrated parameter files, the restart files, the spectral output files and the WAM_Prot file. Map_to_rast creates the HDF5 file of the datasets that SMS can read.

SMS will automatically run all grids in a simulation, even nested grids. The model wrapper will run a process for a coarse grids before running the same process for one of the grids nested inside of it.

Nested Grids

- Used to make a finer grid of a particular area that will use input information from the coarse grid.
- To create a nested grid, make a grid frame around the area of the coarse grid to which you wish to make a nested grid. Then, before the nested grid is mapped, check the "Fine Grid" checkbox and select the coarse grid from the "Coarse Grid" combobox. This will change the values for the newly mapped grid, such as the origin and increment, to correspond to cells in the coarse grid.
- The dataset for the wind of the coarse grid may be used for the fine grid. Other datasets, such as for ice and currents, for the fine grid will need to be made in the same manner as the coarse grid, if they are to be used.
- The nested(fine) grid will have to be added to the simulation as well as the parent(coarse) grid.

WAM Wet and Dry Cells

Wet/dry status for WAM cells is determined solely upon the z values (represented as a depth). If the depth is positive, the cell is wet. If the depth is negative, the cell is dry.

Only wet cells may be used as spectra sites.

To display wet/dry cells: Select the WAM grid Display->Display Options Turn on land and ocean cell For best results also turn off contours.

The following operations may change the wet/dry status of a WAM cell since they are changing the depth values.

1. Select 1 or more WAM cells and then change the Z value found at the top.
2. Select and right click a WAM cell. Choose "Interpolate Bathymetry..." Enter a "single value" on the left side and press "Ok"
3. When converting a WAM map coverage to a WAM grid "Map→2DGrid" whatever value is in the "Depth Options" section will be converted.

WAM Grid Options

The Grid Options dialog contains three tabs which are as follows:

General Tab

Model options

- Water depth model – Choosing a shallow depth model specifies the use of the full dispersion equation. Dispersion equation is w^2 = g k tanh(k h) where w is radial frequency, g is acceleration due to gravity, k is the wave number, and h is the water depth. Computes all phase and group celerity where there is depth dependence.
- Refraction model – If running in deep water, specify "Not used." It is not recommend to use if grid resolution is greater than 3-min (0.05 deg) on a spherical grid.
- Breaking – It is recommended that breaking be used when the water depth model is "Shallow." This will implement limited breaking within the WAM model.

Model time steps

- Propagation time step – The propagation time step is controlled by the CFL condition. The CFL condition requires that the numerical solution cannot move faster than the group speed of the waves. It is recommended that the other model time steps and output time steps be multiples of the propagation time step.
- The CFL criteria must be less than 1.0 for WAM to run successfully. The estimated CFL value is shown below the time step. The value shows in green if the settings meet the criteria or red if not. The CFL computation is dependent upon whether deep or shallow water depth model is being used as well as the refraction model. Currently the CFL estimation in SMS does not account for currents but this is used within WAM. Therefore, the CFL estimate may look okay in SMS but fail when running WAM. If this occurs further reduce the propogation time step or change some of the other options.
- Source time step – This is the time step for the source term integration time step. Increasing this value can greatly decrease run times. However, it should be used with caution. It is recommended that the source term time step be no greater than 10 times the propagation time step.

Output time steps

- Spatial datasets – This is the frequency of output that will be generated that can be loaded into SMS as datasets. The WAM file it creates is the "integrated parameter map file."
- Spectra – This is the frequency of output for spectra data at defined observation cells.
- Close/Reopen – When writing files on a computer, often data being written is stored temporarily in RAM before being written to disk. If a file is in this state and WAM crashes, the final data may not be complete. Closing and reopening the output files forces the file system to complete writing the data to file.

Output Tab

The Output tab consists of different Model options.

- Sea waves: waves generated by the wind blowing at the time, and in the recent past, in the area of observation.
- Swell waves: waves which have travelled into the area of observation after having been generated by previous winds in other areas. These waves may travel thousands of kilometres from their origin before dying away. There may be swell present even if the wind is calm and there are no 'sea' waves.

Spatial Inputs Tab

WAM uses spatial input for ice cover, currents, and wind. The Spatial Inputs section of the WAM Model Control is used to manage these datasets.

Ice Cover

Dataset

The ice cover dataset is an editable scalar dataset. After creating the dataset using the Editable Datasets dialog accessed from the Model Control, the dataset will be displayed in the project explorer in a folder named Editable Datasets. Only one ice cover dataset may exist at a time.

File

Rather than using a dataset you can specify the file name for ice.

Currents

Dataset

The currents dataset is specified to be any vector dataset on the current grid. The dataset and time step must both be selected. The selection may be removed by clicking on the Clear button. This unselects the dataset as the specified currents dataset, but it does not remove the dataset from the SMS project.

File

Rather than using a dataset you can specify the file name for currents.

Wind

Dataset

The winds dataset can be set to any vector dataset that belongs to the current grid or another grid whose angle is zero and whose extents cover the current grid. Nested grids may use the wind dataset of the parent grid. The time steps in the dataset used must match those used with the WAM simulation.

File

PBL can be used to create an input wind file for WAM by selecting "WAM input" in the PBL Model Control. The fort.12 file created by PBL may be selected as the wind data. Note: This option is only useful if you have access to the "Planetary Boundary Layer" model which isn't available through SMS.

WAM Simulation Model Control

The WAM Simulation Model Control can be accessed by right-clicking on the Simulation and selecting Model Control from the menu.

General Tab

Starting Frequency Band

This is a very important parameter in WAM. Based on historical testing of the source terms (see Komen et al., 1994) there was a prescribed frequency range to tune the Discrete Interaction Approximation and the other source terms defined in the energy balance equation. This work used a starting frequency band of 0.04177248 Hz. Also the definition of the frequency band range is dictated by:

$f(n+1) = 1.1 * f(n)$where $n = 1, NFRE - 1$($NFRE$is number of frequency bands)

For many application this starting frequency may be too high for the case of a large-oceanic scale domain like the Pacific Ocean where there is the potential for energy carrying frequencies lower than the designated threshold (i.e. 25-sec peak spectral wave periods, where wave period is the inverse of the frequency). Also, for small-scale fully enclosed bodies of water (i.e. the Great Lakes, reservoirs etc.) the range of wave frequencies is much higher, where the lowest frequency may be on the order of 0.0833-Hz (or 12-sec). The initial frequency value can be changed, upward or downward, as long as the range of frequency values can be defined from the 0.0417728-Hz standard. For the example of the Pacific Ocean implementation, a starting value of 0.03138428-Hz is used, where f(4) = 0.0417728-Hz as defined by the above equation. Setting a higher starting point for a Great Lakes simulation and knowledge of the wave climate an initial value could be 0.050545-Hz. Standard practices use conservatism allowing for energy to be transferred. Also, it is also advantageous to set the first frequency band high enough to minimize the computational time, where the propagation of wave energy is dictated by the group speed of the first frequency band. The lower the frequency, the shorter the stable propagation time step. The number of direction bands is nominally set to 24 or a 15-deg directional resolution of the wave spectra. Selection of a higher directional resolution (or increasing the number of directional bins) will linearly increase computational time because the directional loop is the outer-most loop in all of WAM’s calculations.

Simulation run times

The starting and ending time of the simulation must be entered in this section. The run times must be less or equal to the time available for wind data.

Spectra Tab

The parameters in the spectra tab with the exception of Fetch are Jonswap parameters.

Spectra Sites

Since the spectra data of every cell of the grid at each timestep would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting Spectral Energy. The spectra files may be read back into SMS by selecting the grid, and choosing Read Spectral Files from the WAM menu. The spectra files should be within the grid's folder within the simulation folder.

WAM Spectra from STWAVE Grids

Spectra Sites

Since the spectra data of every cell of the grid at each timestep would take an unreasonable amount of disk space, WAM writes spectra data at user specified cells. These cells are referred to as spectra sites and can be specified within the SMS interface. The spectra output can also be visualized on these cells.

Spectra sites can be made at any ocean cell on a Cartesian grid. To make a spectra site, select a cell on the desired Cartesian grid and change the cell attribute to Spectra Site and give it a name. If more than one cell is selected, then the name the spectra sites will receive is the name given concatenated with a number.

After WAM has run, and the spectra files have been read back into SMS, the spectral energy graph of the spectra sites may be viewed by right-clicking on the selected cell where the site is and selecting Spectral Energy. The spectra files may be read back into SMS by selecting the grid, and choosing Read Spectral Files from the WAM menu. The spectra files should be within the grid's folder within the simulation folder.

WAM Spectra From STWAVE Grids

WAM can be used to provide boundary condition information to a spectral wave model such as STWAVE. WAM->STWAVE has a defined and support mechanism for converting WAM spectral output into STWAVE input.

The first part of this process is to define the locations where WAM will generate output for the STWAVE BC. Two or more WAM cells may be used along a STWAVE boundary. The STWAVE grids must exist and the boundaries types specified (which side(s) to apply spectra.

To create spectral sites along the STWAVE boundaries:

1. Verify/Modify the boundary types of a STWAVE Grid.
   a. Select a STWAVE grid.
   b. Select menu item "STWAVE" → "Model Control" then press the "Boundary Condition..." button.
   c. Select the appropriate boundary type for each edge.
   d. Repeat for each STWAVE grid.

1. Select and right-click on a WAM grid from the tree item on the left of SMS.
2. Select "Create spectral sites..." to bring up the "WAM spectral spectral sites from STWAVE" dialog.
3. Select the grids you would like to generate sites for.
4. Select a spacing option on the right hand side and the number of points/sites.
5. Press "Ok".

Each STWAVE boundary cell will have a candidate WAM corner (where WAM spectra are output). A candidate corner is the nearest WAM corner to that STWAVE boundary cell. Multiple STWAVE cells may have the same candidate WAM corner. Duplicate WAM corners are automatically removed.

Spacing can be done by either spacing along candidate corners or by specifying an exact number of sites/corners along each boundary.

If spacing by candidate point(s) is chosen:

- Spacing by every 1 candidate points will keep all candidate corners and make them spectra sites.

- Spacing by every 2 candidate points will keep every other corner and make those spectra sites.

Regardless of the option selected at least two corners will be used on each STAVE boundary.

6. Riverine and Estuarine Models

HYDRO AS-2D

HYDRO AS-2D	
Model Info	
Model type	2D current, pollutant, and sediment transport simulation.
Developer	Dr.- Ing. Marinko NUJIC
Web site	HYDRO AS-2D web site [1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

HYDRO AS-2D performs 2D modeling of bodies of water. The procedure integrated in HYDRO AS-2D is based on the numerical solution of the 2D current equations with Finite-volume-Discretization. In addition to 2D current simulation, HYDRO AS-2D can also simulate pollutant and sediment transport.

Functionality

Features

- Solves hundreds of thousands of elements very quickly
- Calculation accuracy guaranteed by extensive lab tests and real-world applications
- High stability, robustness and exactness for a wide spectrum of hydrodynamic conditions
- Volumetric accuracy of the tidal wave propagation on complicated topography
- Solutions showing flow speed, direction, and flood depth
- The bed changes as a result of sediment erosion and sediment deposition are modeled
- Fully coupled transport and hydrodynamic solver ensures current changes due to sediment scour/deposition are accurately simulated
- Simulate transport of up to five grain dimensions at the same time

Applications

- River flooding analysis
- Sediment erosion/depositions studies in river channels and floodplains
- Tidal wave propagation
- Pollutant dispersion in a waterway

Graphical Interface

HYDRO AS-2D uses the Generic Model Graphical Interface.

External Links

- PowerPoint presentation of a Dam Break Study [2]
- Hydrotec HYDRO AS-2D website (in German) [3]
- Hydrotec HYDRO AS-2D website (in German) [4]
- HYDRO AS-2D model developer website (in German) [1]

Related Topics

- Generic Model Interface

References

[1] http://ib-nujic.de/
[2] http://www2.uni-wuppertal.de/FB11/lehrgebiete/igaw/docs_oertel/Two-dimensional_flow_simulations.pdf
[3] http://www2.hydrotec.de/unternehmen/hydrothemen/hydrothemen07/hydro-as-2d/
[4] http://www2.hydrotec.de/vertrieb/hydro_as_2d/

RIVERFLO-2D

RiverFLO-2D	
Model Info	
Model type	2D current and sediment transport simulation.
Developer	Dr. Reinaldo Garcia
Web site	[1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation

RiverFLO-2D v3 is a hydrodynamic, mobile bed model for rivers and estuaries that uses a stable and powerful finite element method to compute high resolution flood hydraulics, including supercritical and subcritical flows over dry or wet river beds. Use of flexible triangular mesh allows resolving the flow field, sediment transport and bed elevation changes around key features in complex river environments. RiverFLO-2D has been applied on numerous river projects worldwide including large rivers in the United States, South America, Europe and Asia.

Functionality

Features

- Flexible triangular finite element meshes
- Mesh refinement to resolve complex river features
- Supercritical and subcritical flow
- Steady and time-dependent flow
- Mobile-bed sediment transport
- Numerically stable explicit solution scheme
- Dry-wet bed capability not requiring initially wet elements
- Parallelized code for multiple-core processor computers
- Documented input ASCII, free format files
- Comprehensive ASCII output files
- Graphical output in a variety of formats

Using the Model / Practical Notes

The RiverFLO-2D installation package includes several executable files. To connect SMS to the numeric engine:

1. Select the *Preferences* command from the *Edit* menu
2. Click on the "*File Locations* tab in the SMS Preferences dialog
3. In the *Model Executables* area, scroll down to the *Generic* line and click on the button on the right side. A dialog to select the executable will appear.
4. Browse to the area where SMS is installed (the default is in "Program Files\SMS 11.1 ##-bit"). From there browse into the "models\RiverFLO-2D" folder.
5. Select the executable file named "RiverFLO-2Dm3.exe". This is the engine itself that SMS will launch to analyze a simulation.
6. Click the *Open* button to select the file and the *OK* button to close the preferences dialog.

Graphical Interface

RiverFLO-2D uses the Generic Model Graphical Interface which allows you to construct the grid, assign model parameters and boundary conditions based on a template provided by the RiverFLO-2D developers, and view the output of an analysis.

Modeling hints for RiverFLO-2D

- In the model parameters, RiverFLO-2D supports a *Variable Timestep* option. When this is on, the user must specify a multiplier. Our experience has shown that you may need to try various values for this multiplier in order to get proper mass conservation.
- The global parameter group *Graphic Output Options* causes the engine to launch a separate window during analysis to provide graphical feedback of its progress.

Reading Legacy Projects into SMS

An existing RiverFLO-2D project consists of the following files:

- *.dat – basic input data
- *.fed – geometry
- *.ifl – boundary conditions in open countours
- *.hvt – water surface elevation vs. time
- *.qvt – flow rate vs. time
- *.plt – plot options
- *.sed – sediment data
- *.tba – list of boundary nodes

Aquaveo does not currently have a utility to convert these files directly to SMS format. In order to work with data from this project in the SMS environment follow these steps:

Mesh Data

- Open the *.fed file in a file editing program such as Notepad ++. Copy and paste the data into Microsoft Excel.
- In Excel, click on the data menu and select 'Text to Columns'. Then select 'Fixed Width'. This will divide the data into individual columns. Click 'next' twice, then 'finish'.
- We are only concerned with the data in the first three columns so delete the data in the other columns.
- We will delete one more sets of data now. Scroll down and you will notice that the data changes. This is because it switches from mesh data to element data. Delete the element data so you only have the mesh data.
- Do a file 'Save as' and save the file as a comma delimited (*.csv) file. Click Ok, then click 'No'. The file is now ready to be read into SMS.
- When reading the file into SMS, use the Import Wizard. In the Import Wizard, make sure the file is set to 'Delimited' and click next. Change the 'SMS data type' to 'mesh', then click 'finish'.
- Once read into SMS, delete the undesired elements.

Nodestring Boundary Conditions

- Open the .hvt file in a file editor and copy and paste the data into Excel. Click on the data menu and select 'Text to Columns'. Then select 'Fixed Width'. Click 'next' twice, then 'Finish'.
- Repeat the above step for the *.qvt file. The data in these files are now ready to be copied and pasted into SMS and assigned as boundary conditions in their respective nodestrings.

Control Data

The *.dat file does not need to be converted

- Open the *.dat file into the Argus One RiverFLO-2D engine. Note the data that appears.

- In SMS, click on the RiverFLO-2D menu and click 'Global Parameters'. Click on the 'Control Data' tab and record the data from the *.dat file in the RiverFLO-2D engine.

Remaining Files

These files are also read into the Argus One RiverFLO-2D engine, and the data from there can be manually recorded in SMS under their respective tabs in RiverFLO-2D → Global parameters.

Case Studies / Sample Problems

Hoh River Washington

This tutorial is distributed with RiverFLO-2D and includes a typical riverine application and floodplain.

Related Topics

- Generic Model Interface

References

[1] http://hydronia.net/computer-models-software/riverflo-2d/

SRH-2D

SRH-2D	
Model Info	
Model type	Two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems
Developer	Yong Lai Bureau of Reclamation
Web site	SRH-2D web site [1]
Tutorials	General Section • Data Visualization • Mesh Editing • Observation Models Section • SRH-2D (Pending)

About SRH-2D

SRH-2D, Sedimentation and River Hydraulics – Two-Dimensional model, is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems under development at the Bureau of Reclamation. Different versions of SRH-2D contain different modules, as listed below (only version 2 is released and the rest will be released in the future):

- SRH-2D version 2: Modeling of flow hydraulics for stream/river systems
- SRH-2D version 3: Mobile bed sediment transport module added to v.2 **(Not yet released)**
- SRH-2D version 4: Temperature and vegetation module added to version v.2 **(Not yet released)**
- For watershed runoff modeling, SRH-W v.1.1 should be used.

About SRH-2D Version 2

SRH-2D version 2 solves the 2D dynamic wave equations, i.e., the depth-averaged St. Venant equations. Its modeling capability is comparable to some existing 2D models but SRH-2D claims a few boasting features. First, SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. In practice, the hybrid mesh of quadrilateral and triangular cells is recommended though purely quadrilateral or triangular elements may be used. A hybrid mesh may achieve the best compromise between solution accuracy and computing demand. Second, SRH-2D adopts very robust and stable numerical schemes with a seamless wetting-drying algorithm. The resultant outcome is that few tuning parameters are needed to obtain the final solution. SRH-2D was evolved from SRH-W which had the additional capability of watershed runoff modeling. Many features are improved from SRH-W.

Major Features of SRH-2D v. 2

Major SRH-2D capabilities are listed below

- 2D depth-averaged dynamic wave equations (the standard St. Venant equations) are solved with the finite-volume numerical method
- Steady state (with constant discharge) or unsteady flows (with flow hydrograph) may be simulated
- An implicit scheme is used for time integration to achieve solution robustness and efficiency
- An unstructured arbitrarily-shaped mesh is used which includes the structured quadrilateral mesh, the purely triangular mesh, or a combination of the two. Cartesian or raster mesh may also be used. In most applications, a combination of quadrilateral and triangular meshes is the best in terms of efficiency and accuracy
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need for special treatments
- Robust and seamless wetting-drying algorithm; an
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output variables include the above, plus Froude number, bed shear stress, critical sediment diameter, and sediment transport capacity.

SRH-2D is a 2D model, and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with side channel and agricultural returns, and with braided channel systems. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns, flow recirculation, lateral velocity variation, and flow over banks and levees.

Features

- SRH-2D solves the 2D depth-averaged form of the diffusive wave or the dynamic wave equations. The dynamic wave equations are the standard St. Venant depth-averaged shallow water equations
- Both the diffusive wave and dynamic wave solvers use the implicit scheme to achieve solution robustness and efficiency
- Both steady or unsteady flows may be simulated
- All flow regimes, i.e., subcritical, transcritical, and supercritical flows, may be simulated simultaneously without the need of a special treatment
- Solution domain may include a combination of main channels, side channels, floodplains, and overland
- Solved variables include water surface elevation, water depth, and depth averaged velocity. Output information includes above variables, plus flow inundation, Froude number, and bed shear stress

SRH-2D is a 2D model and it is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, with perched rivers, with multiple channel systems, and with complex floodplains. A 2D model may also be needed if one is interested in local flow velocities, eddy patterns and flow recirculation, lateral variations, flow spills over banks and levees, and flow diversion and bifurcation.

The Bureau of Reclamation does not provide technical support for SRH-2D.

Graphical Interface

SRH-2D uses the Generic Model Graphical Interface. The SRH-2D version 2.0 Distribution [2] includes SRH2D template files for both SMS 8.0 and SMS 10.0.

External Links

- Bureau of Reclamation SRH-2D Website [1]

External Links – SRH-2D Version 2.0

- SRH-2D version 2.0 Theory and User Manual [3]
- SRH-2D version 2.0 Distribution [2] – Includes Software, Manual, Template file for integration into SMS interface, and Tutorials

Papers / Presentations

- SRH-2D Theory Paper [4]
- SRH-2D Training Presentation [5]
- 2005 US-China Workshop Paper [28][29]
- 2006 FISC Paper on Savage Rapids Dam Removal Project - "Comparison of Numerical Hydraulic Models Applied To The Removal of Savage Rapids Dam Near Grants Pass, Oregon" [5]
- FISC 2006 Short Course Presentation [16]
- Using Bathymetric LiDAR and a 2-D Hydraulic Model to Identify Aquatic River Habitat [6]
- List of journal articles using SMS by Prof. Greg Pasternack, UC Davis [2]

Project Reports

- Bountry J.A. and Lai, Y.G. (2006)."Numerical modeling of flow hydraulics in support of the Savage Rapids Dam removal." [21]
- Lai, Y.G., Holburn, E.R., and Bauer, T.R. (2006)."Analysis of sediment transport following removal of the Sandy River Delta Dam." [20]
- Lai, Y.G. and Bountry, J.A. (2006). "Numerical hydraulic modeling and assessment in support of Elwha Surface Diversion Project." [19]
- Lai, Y.G. and Bountry, J.A. (2007). "Numerical modeling study of levee setback alternatives for lower Dungeness River, Washington" [11]

In the News

- Wired.com article "Computer Modeling Smoothes a Dam Hard Job" [7]
- Photos related to Wired.com article "Computer Modeling Smoothes a Dam Hard Job" [8]

External Links – SRH-W

SRH-W Version 1.1

- SRH-W version 1.1 User Manual [9]
- SRH-W version 1.1 Distribution Package [10] – Includes Software, Manual, and Tutorials

Papers / Presentations

- 2006 FISC Watershed Modeling Paper - Watershed Simulation with an Enhanced Distributed Model [11][12]

Related Topics

- Generic Model Interface

References

[1] http://www.usbr.gov/pmts/sediment/model/srh2d/index.html
[2] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/SRH2D-v2-Distribution-Package-Jan2009.zip
[3] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Manual-SRH2D-v2.0-Nov2008.pdf
[4] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Theory_Paper-SRH2D-v2-2009.pdf
[5] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2006-Savage-Rapids-FISC-paper.pdf
[6] http://www.usbr.gov/pmts/sediment/projects/Yakima/download/EWRIpaper_final_011307.pdf
[7] http://www.wired.com/science/discoveries/news/2007/07/dam
[8] http://www.wired.com/science/discoveries/multimedia/2007/07/dam_gallery?slide=1&slideView=10
[9] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/SRH-W%20v1.1%20User%20Manual%20June2007.pdf
[10] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/SRH-W1.1-Distribution-Package-2007.exe
[11] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/2006-3rd-FIHMC-Paper-SRH-W.pdf
[12] http://www.usbr.gov/pmts/sediment/model/srh2d/Downloads/Paper%20Watershed%20FISC%202006.pdf

Steering

The steering tool has been added to facilitate the process of launching models multiple times. To launch the steering tool, choose the menu command *Data | Steering Module*.

The steering tool can be used for single model (spin down) simulations of RMA2 and FESWMS.

The steering tool can also be used to facilitate the transfer of data from wave models to circulation models and back. Currently, ADCIRC / CMS-Wave and CMS-Flow / CMS-Wave links are supported.

Related Topics

- SMS Models
- RMA2 Spindown
- FESWMS Spindown
- ADCIRC / CMS-Wave Steering
- CMS-Flow / CMS-Wave Steering

NOTE: CMS-Flow (formerly known as M2D) and CMS-Wave (formerly known as WABED) are components of the Coastal Modeling System (CMS).

External Links

- Jun 2002 ERDC/CHL CHETN-IV-41 SMS Steering Module for Coupling Waves and Currents, 1: ADCIRC and STWAVE [35]
  - Please see this forum post [2] for an explanation of ADCIRC and STWAVE steering
- Jun 2002 ERDC/CHL CHETN-IV-42 Coupling of Regional and Local Circulation Models ADCIRC and M2D (now know as CMS-Flow) [1]
- Dec 2003 ERDC/CHL CHETN-IV-60 SMS Steering Module for Coupling Waves and Currents, 2: M2D (now know as CMS-Flow) and STWAVE [32]

6.1. ADH

ADH

"ADH is a state-of-the-art ADaptive Hydraulics Modeling system developed by the Coastal and Hydraulics Laboratory (CHL), ERDC, USACE [1], and is capable of handling both saturated and unsaturated groundwater, overland flow, three-dimensional Navier-Stokes flow, and two- or three-dimensional shallow water problems." [2]

Information about the ADH model can be found at the ADaptive Hydraulics Modeling HomePage [2].

Graphical Interface

Dialogs

SMS provides the capability to view and edit model parameters via dialog boxes specific to the ADH interface. The following are descriptions of important windows accessible from the ADH menu and right-click menus.

- Model Control
- Boundary Condition Assignment
- Material Properties
- Hot Start Initial Conditions
- Model Check
- Boundary Layers
- Run Model

Files

An ADH simulation is defined using three main files: mesh geometry (*.3dm), boundary conditions (*.bc), and hot start conditions (*.hot). ADH expects a 3-dimensional mesh geometry since it can solve 3D models, but SMS will only interface the 2D capabilities and will therefore only read and write 2D entities in the *.3dm file. A generic SMS mesh geometry file (*.2dm) can be used by ADH if the extension is changed to *.3dm. To facilitate handling the ADH files, SMS creates a simulation file (*.sim) which lists the individual files. Opening the *.sim will load the ADH files in SMS.

Each file is a card based text file that can be viewed and edited by a text editor (i.e. Notepad).

- Mesh geometry file (*.2dm/*.3dm)
- Boundary condition file (*.bc)
- Hot start file (*.hot)
- Boat definition file (*.bt)

Using the Model / Practical Notes

ADH uses string structures to assign properties to entities. The following table describes the relation between the string structures and SMS structures.

ADH to SMS (Reading)

ADH String Structure		SMS Structure
NDS with DB card	and only 1 node ID	Node with boundary condition (BC)
	and 2 or more node IDs	Nodestring with BC
EGS with NB card (and possibly FLX card)		Nodestring with BC (and flux output)
MDS with FLX card		Nodestring with flux output
MTS with MP cards		Material with properties

SMS to ADH (Writing)

SMS Structure			ADH String Structure
Node with BC (Dirichlet)			NDS (only 1 node ID) with DB card
Nodestring	with Natural BC	with flux output	EGS with NB and FLX cards
			EGS with NB card
	with Dirichlet BC	with flux output	NDS (2 or more node IDs) with DB card, and MDS with FLX card
			NDS (2 or more node IDs) with DB card
	with flux output		MDS with FLX card
Material with properties			MTS with MP cards

Case Studies / Sample Problems

The following ADH simulation files have been provided by CHL [1] to test ADH and the SMS interface capabilities. Additional test cases will be posted here as the interface can successfully read, write, and run the ADH model from within SMS. SMS 10.1 Development version using ADH rev #3669 dated August 2007:

- Bayou_Sorrel [3] – Uses the NB OVL and NB OTW boundary condition cards. Includes two materials on a complex mesh.
- Pool5 [4] – Uses the NB OVL and NB OTW boundary condition cards. Includes two inflows with one outflow and four materials on a complex mesh.
- nb dis [5] – Tests the NB DIS boundary card on a straight flume.
- op bt [6] – Tests the OP BT card which includes the vessel movement library. This test case contains a boat definition file (*.bt).
- db lde [7] – Tests the DB LDE pressure lid card on a straight flume.
- db ldh [8] – Tests the DB LDH pressure lid card on a straight flume.
- db lid [9] – Tests the DB LID pressure lid card on a straight flume.

References

[1] http://chl.erdc.usace.army.mil/
[2] https://adh.usace.army.mil/
[3] http://sms.aquaveo.com/ADH-bayousorrel.zip
[4] http://sms.aquaveo.com/ADH-pool5.zip
[5] http://sms.aquaveo.com/ADH-nb_dis.zip
[6] http://sms.aquaveo.com/ADH-op_bt.zip
[7] http://sms.aquaveo.com/ADH-db_lde.zip
[8] http://sms.aquaveo.com/ADH-db_ldh.zip
[9] http://sms.aquaveo.com/ADH-db_lid.zip

ADH 2D Shallow Water Flow

Go to SMS:Models page

"ADH is a state-of-the-art ADaptive Hydraulics Modeling system developed by the Coastal and Hydraulics Laboratory (CHL), ERDC, USACE [1], and is capable of handling both saturated and unsaturated groundwater, overland flow, three-dimensional Navier-Stokes flow, and two- or three-dimensional shallow water problems." [2]

Information about the ADH model can be found at the ADaptive Hydraulics Modeling HomePage [2].

Functionality

Top of page

Graphical Interface

The model was initially included in the 9.2 version release and was overhauled for the 10.0 version release, however it is available only to the model developers. We are continuing to develop an interface to allow SMS users to utilize ADH for 2D shallow water flow and sediment transport analysis.

Files

An ADH simulation is defined using three main files: mesh geometry (*.3dm), boundary conditions (*.bc), and hot start conditions (*.hot). ADH expects a 3-dimensional mesh geometry since it can solve 3D models, but SMS will only interface the 2D capabilities and will therefore only read and write 2D entities in the *.3dm file. A generic SMS mesh geometry file (*.2dm) can be used by ADH if the extension is changed to *.3dm. To facilitate handling the ADH files, SMS creates a simulation file (*.sim) which lists the individual files. Opening the *.sim will load the ADH files in SMS.

Each file is a card based text file that can be viewed and edited by a text editor (i.e. Notepad).

- Mesh geometry file (*.2dm/*.3dm)
- Boundary condition file (*.bc)
- Hot start file (*.hot)
- Boat definition file (*.bt)

Dialogs

SMS provides the capability to view and edit model parameters via dialog boxes specific to the ADH interface. The following are descriptions of important windows accessible from the ADH menu and right-click menus.

- Model Control
- Boundary Condition Assignment
- Material Properties
- Hot Start Initial Conditions
- Model Check (currently unavailable - under development)
- Boundary Layers (currently unavailable - under development)
- Run Model

Top of page

Using the Model / Practical Notes

- ADH String Structures
- Sediment Transport and Bed Layers
- Time Series
- Wind Stations

Top of page

Case Studies / Sample Problems

The following ADH simulation files have been provided by CHL [1] to test ADH and the SMS interface capabilities. Additional test cases will be posted here as the interface can successfully read, write, and run the ADH model from within SMS.

SMS 10.1 Development version using ADH rev #3669 dated August 2007:

- **Bayou_Sorrel** [3] - Uses the NB OVL and NB OTW boundary condition cards. Includes two materials on a complex mesh.
- **Pool5** [4] - Uses the NB OVL and NB OTW boundary condition cards. Includes two inflows with one outflow and four materials on a complex mesh.
- **nb dis** [5] - Tests the NB DIS boundary card on a straight flume.
- **op bt** [6] - Tests the OP BT card which includes the vessel movement library. This test case contains a boat definition file (*.bt).
- **db lde** [7] - Tests the DB LDE pressure lid card on a straight flume.
- **db ldh** [8] - Tests the DB LDH pressure lid card on a straight flume.
- **db lid** [9] - Tests the DB LID pressure lid card on a straight flume.

Top of page

Go to SMS:Models page

References

[1] http://chl.erdc.usace.army.mil
[2] https://adh.usace.army.mil

ADH Bed Layers Assignment

To open this window, global bed layers must be specified in Model Control on the Global Material Properties tab and at least one node must be selected. After a selection is made, select the ADH | Assign Bed Layers... menu item or use the right-click menu.

Dialog Description

The following controls specify the bed layer specifications to be assigned to the currently selected nodes to overwrite the global specifications. Overwriting is specified on a layer by layer basis at the selected nodes. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data.

Bed Layers (untitled group box)

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. The specification rows are as follows:

- Edit differing specifications – Contains Edit check boxes and only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. A check box is only placed in the columns of layers with conflicts (denoted by asterisks appended to the layer ID). The entire layer column will be disabled unless Edit is checked. If it is checked when accepting changes (exiting the window by clicking OK) the specifications for the bed layer will be applied to all selected nodes. Bed Layers that are not edited (remain unchecked) will not be changed.
- Overwrite global specification – Contains Overwrite check boxes for each bed layer to specify whether the layer will be overwritten. If checked, the parameter cells of the column are enabled for editing, else the global specification values of the bed layer are displayed as read-only. The check box is disabled if a layer conflict exists and Edit is unchecked.
- Sediment distribution – Title cell with each defined sediment listed below in its own row. The row contains Normalize buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%. The button is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Sediment (name) – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Total – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%. The cell is disabled if a layer conflict exists and Edit is unchecked.
- Thickness – Consists of the each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the MP SBN card. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Bulk Density – Consists of each bed layer's bulk density measured in kilograms per cubic meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.

- Erosion crit. shear – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Erosion rate constant – Consists of each bed layer's unitless erosion rate constant. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.
- Erosion rate exponent – Consists of each bed layer's unitless erosion rate exponent. The value must be a positive non-zero real number and correlates to the MP CBN card. This row only appears if a cohesive sediment (clay or silt) is defined. The cell is disabled if a layer conflict exists and Edit is unchecked and read-only or enabled if Overwrite is unchecked or checked, respectively.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Asterisks (*) denote bed layers with specifications that differ between the selected nodes. This note only appears when multiple nodes are selected and the bed layers specified at the nodes conflict. Asterisks are appended to the layer ID of differing layers in the title row across the top of the spreadsheet.

Go to SMS:ADH page

ADH Boat Definition File Cards

The following lists ADH *.bt cards that are recognized by SMS. For a description of card specifics, please use the ADH Quick Reference [1].

Boat Properties

	Read	Edit	Write	Notes
BLEN	Allowed	Allowed	Allowed	
BOAT	Allowed	Allowed	Allowed	
BWID	Allowed	Allowed	Allowed	
CBOW	Allowed	Allowed	Allowed	
CSTR	Allowed	Allowed	Allowed	
DRFT	Allowed	Allowed	Allowed	
ENDD	Allowed	Allowed	Allowed	
FDEF	Allowed	Allowed	Allowed	
PBOW	Allowed	Allowed	Allowed	
PROP	Allowed	Allowed	Allowed	
PSTR	Allowed	Allowed	Allowed	
SDEF	Allowed	Allowed	Allowed	

Go to SMS:ADH page

References

[1] https://adh.usace.army.mil/ADH_Quickref/

ADH Extract WSE

SMS now has the ability to extract water surface elevation to a nodestring boundary condition from an existing dataset. This is done by right-clicking on the nodestring and selecting Boundary Condition → Extract Water Surface Elevation... and selecting the dataset to extract from. SMS will break up the existing nodestring into segments and extract the water surface elevation at the midpoint of each segment. The nodestrings are then assigned the Tailwater (outflow) boundary condition and the extracted data is set as the time series. This data can be edited in the ADH Boundary Condition Assignment dialog.

Go to SMS:ADH page

ADH Hot Start

ADH Hot Start (*.hot) File

The hot start file is mandatory and contains initial conditions for each node of a mesh. ADH allows for the specification of depth and velocity hydrodynamics, and sediment concentration and displacement datasets. The model requires at least depth initial conditions for hydrodynamics, and non-specified conditions will be defaulted to values of zeros (0.0).

The *.hot file format is actually the same as the ASCII dataset Files (*.dat) but necessitates the special extension since it should only contain following specifically named datasets:

Hot Start Datasets

Name	Initial Condition
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity
"ioc" or "IOC"	Sediment concentration (SMS currently does not support this)
"iod" or "IOD"	Sediment displacement (SMS currently does not support this)
Name	**Initial Condition**
"ioh" or "IOH"	Depth
"iov" or "IOV"	Velocity

SMS will recognize and read in a hot start file if the dataset definition matches the current mesh definition (number of nodes and elements) and the ADH is the active model (other models also use a *.hot extension). Valid datasets with the above names are added to a locked "ADH Hot Start" folder in the Project Explorer. The folder and datasets can not be deleted or renamed from the tree. See Hot Start Initial Conditions for interface information.

Go to SMS:ADH page

ADH Hot Start Initial Conditions

Go to SMS:ADH page

This window can be accessed by the ADH | Hot Start Initial Conditions menu item. ADH requires that at least an initial depth dataset is defined to start a simulation. ADH can not start with a completely dry domain.

The initial condition datasets will be generated and appear in the "ADH Hot Start" folder of the Project Explorer. This folder and the datasets are locked for editing except from this dialog.

Dialog Description

Depth

Constant water surface / Constant depth / Use defined dataset radio group specifies the method for defining the initial depth dataset. The *Constant water surface* control group includes the *Elevation* real number edit field which only allows elevations greater than the maximum mesh elevation. This produces a fully flooded domain with a flat horizontal water surface. The *Constant depth* control group similarly includes the *Depth* positive real number edit field. The depth value is applied to every node to create a water surface which is offset from the mesh geometry. The *Use defined dataset* control group allows for the selection of an existing scalar dataset located in the Project Explorer via *Select Dataset...* button. The name and time step information (if transient) of the selected dataset will appear below this button.

Velocity

Specify velocity check box specifies whether an initial velocity dataset will be included and if checked, enables the *Constant velocity / Use defined dataset* method specification radio group. The *Constant velocity* control group includes *X* and *Y* velocity component real number edit fields. The composite velocity vector will be applied to every node in the domain. The *Use defined dataset* control group allows for the selection of an existing vector dataset located in the Project Explorer via *Select Dataset...* button. The name and time step information (if transient) of the selected dataset will appear below this button.

Go to SMS:ADH page

ADH Material Properties

Go to SMS:ADH page

To open this window, select the ADH | Material Properties... menu item.

Dialog Description

Materials (Untitled group box)

Name text displays the name of the currently selected material in the material list.

ID text displays the numeric ID of the currently selected material in the material list.

List box contains the existing globally defined materials.

ADH Material Properties (Tab control)

This tab control contains the following three tabs:

Properties

Manning's n roughness real number edit field limited to a value of 0.0 to 0.1. This correlates to a FR MNG card applied to a material.

Kinematic eddy viscosity / Estimated eddy viscosity radio group specifies which method to use and correlates to the MP EVS and MP EEV cards, respectively. The *Kinematic eddy viscosity* control group includes the eddy viscosity tensor edit fields of *EVxx*, *EVyy*, and *EVxy*. The *EVxy* is specified once, and copied to the second term. All edit fields allow positive real number input. The *Estimated eddy viscosity* includes the *Weighting factor* real number field. This factor is used with other model conditions to calculate the eddy viscosity during the simulation.

Include coriolis force check box and the associated *Latitude* edit field will include the MP COR when checked. The edit field allows real number input between -90.0 and 90.0.

Exclude from computations (remove from domain) check box correlates to a OFF card applied to a material.

Meshing and Boundary Conditions

Mesh Refinement

Maximum number of refinement levels positive integer edit field correlates to the MP ML card.

Error tolerance for refinement positive real number edit field correlates to the MP SRT card.

Boundary Condition Assignment

Apply rain or evaporation check box with the associated *Flow (per unit area)* curve button, *Series time units* combo box, and *Specify curve fir tolerance* check box and positive integer edit field defines the rain/evaporation series. Clicking on the curve button will open the XY Series Editor. This control group correlates to the NB OVL boundary condition card with a XY1 series definition card.

Transport Constituents

List box contains the existing transport constituents.

Molecular diffusion rate edit field displays the parameter value for the currently selected transport constituent. This data pertains to the MP DF card.

Error tolerance for refinement edit field displays the parameter value for the currently selected transport constituent. This data pertains to the MP TRT card.

Miscellaneous (Outside of any group)

General Material Properties... button opens the Materials Data window. Any changes made to global materials will be reflected in the materials list box.

Go to SMS:ADH page

ADH Run Model

Go to SMS:ADH page

To access the ADH model wrapper, select the ADH | Run Model menu item.

If the simulation has been changed and not yet saved, you will be prompted to do so before running the model. You will also be prompted to locate the Pre-ADH and ADH executables prior to running each if SMS does not know the location or find the executable. Model executable locations can be managed in on the *File Locations* tab of the Preferences Dialog, accessed from the Edit | Preferences... menu item.

Pre-ADH takes the simulation input files (*.3dm, *.bc, and *.hot), performs some checks and creates the *.adh model input file. ADH reads the model input file, performs calculations and writes solution files (*.dat).

Simulation Input	**Pre-ADH**	**Model Input**	**ADH**	**Solution Output**
Geometry (*.3dm)		ADH Input		Solution Data Sets
Boundary Condition (*.bc)		(*.adh)		(*.dat)
Hot Start (*.hot)				

If Pre-ADH is not successful (the *.adh file is not written) then SMS will not allow ADH to be executed. Upon completion of ADH, the solution files can be automatically loaded into SMS or opened manually.

Dialog Description

Progress

Status line is blank for Pre-ADH and provides time step computation information for ADH.

Progress bar and text shows the percentage of completion.

Run Time displays the time since starting the executable. This is updated when the progress is updated.

Output

Pre-ADH Output or *ADH Output* field displays the last 1000 lines of commentary given by the running executable.

Output Text To File allows all commentary provided by the executable to be saved to a text file (*.txt). This button is enabled upon completion of the executable.

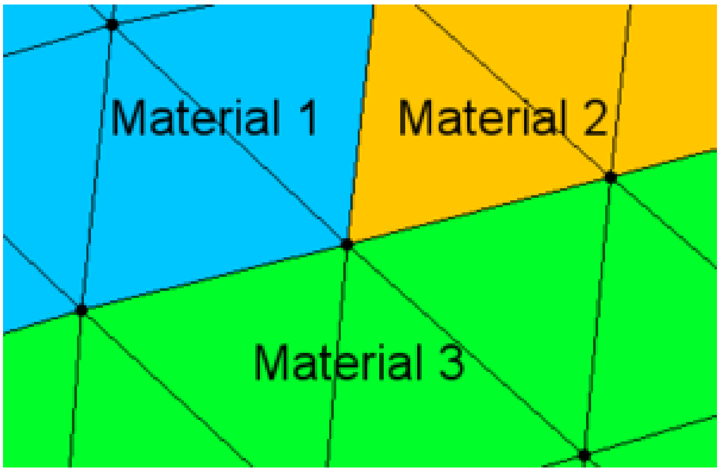
Load Solution check box is available only for ADH and is enabled upon successful completion of ADH. Check this option to automatically load the solution data sets into SMS.

Abort / *Run ADH* / *Exit* button controls the progress of the model wrapper.

Go to SMS:ADH page

ADH Sediment Transport and Bed Layers

Sediments properties defined on the CN CON, CN SND, and CN CLA cards describe the properties of the sediment particle when it is first deposited. Bed layers specified for a simulation are previously deposited layers defined by the MP SBA, MP SBM, and MP SBN cards. While running ADH, the previously deposited layers are aggraded and degraded and the solution datasets provide the resulting bed profile conditions during the simulation. For sediment transport with cohesive sediments, bed layers also have cohesive layer attributes defined by the MP CBA, MP CBM, and MP CBN cards to account for settlement and compaction of the previously deposited layer.



Material Bed Layer Specification Conflict

ADH provides the MP SBM and MP CBM cards to apply a bed layer specifications to nodes (which overwrites the global specifications) by specifying a material, however conflicts can arise since material type is an attribute of an element and not a node. As depicted in the image, a node can define the boundary between multiple materials and if each material has an associated bed layer specification, the central node can have three different specifications. ADH will assign material bed layer specifications by applying them to nodes and overwritting prevoius specifications in the order the cards are read from the *.bc file. Therefore, if MP SBM (or MP CBM) cards for material 1, material 2, and material 3 are read in that order, the central node (in the picture) will finish with the specification for material 3. SMS will read the *.bc file by the same process, however, bed layer specifications by material type will not be allowed in the interface (in Material Properties) and not written to file since materials are not currently prioritizable. Bed layers can be specified to nodes using a material type by selecting the Edit | Select by Material Type... menu item with the Select Node tool active, but the material type is only used to filter the selection and is not associated to the actual node or bed layer specification. SMS will only write MP SBA, MP CBA, MP SBN, and MP CBN cards to file. Bed layers are in continuous ascending ordered by ID starting with 1 as the deepest specified layer in the profile and the highest numbered layer at the top, exposed to the stream flow. ADH automatically adds a solid boundary, consisting completely of an extra immobile (non-erodable) sediment, below Layer 1 which is not presented in the interface but will be included in solution files.

ADH Bed Layering Order

Stream Flow
Layer X
...
Layer 3
Layer 2
Layer 1
Solid Boundary

Bed layers specifications can be edited using the following:

- ADH | Model Control... menu item opens the Model Control window where global bed layers are specified on the Global Material Properties tab.
- ADH | Assign Bed Layers... menu item opens the Bed Layers Assignment window for specifying bed layers at the nodal level. Available only with a current node selection.
- Select Node tool Assign Bed Layers... right click menu item opens the Bed Layers Assignment window for specifying bed layers at the nodal level. Available only with a current node selection.
- Select Node tool Delete Bed Layers right click menu item deletes the nodal specification which overwrites the global. Available only when the current node selection includes at least one nodal bed layer specification.
- Select Node tool Find Equivalent Bed Layers... right click menu item selects all nodes of the mesh that are equivalent to a bed layer specification of the current node selection.

Go to SMS:ADH page

ADH String Structures

ADH uses string structures to assign properties to entities. The following table describes the relation between the string structures and SMS structures.

ADH to SMS (Reading)

ADH String Structure		**SMS Structure**
NDS with DB card	and only 1 node ID	Node with boundary condition (BC)
	and 2 or more node IDs	Nodestring with BC
EGS with NB card (and possibly FLX card)		Nodestring with BC (and flux output)
MDS with FLX card		Nodestring with flux output
MTS with MP cards		Material with properties

SMS to ADH (Writing)

SMS Structure			ADH String Structure
Node with BC (Dirichlet)			NDS (only 1 node ID) with DB card
Nodestring	with Natural BC	with flux output	EGS with NB and FLX cards
			EGS with NB card
	with Dirichlet BC	with flux output	NDS (2 or more node IDs) with DB card, and MDS with FLX card
			NDS (2 or more node IDs) with DB card
	with flux output		MDS with FLX card
Material with properties			MTS with MP cards

Go to SMS:ADH page

ADH Time Series

ADH includes several model specific time series (curve) groups to organize model curves by type. The curves of these groups can be viewed and edited in the Time Series editor and include time series attributes specific to ADH. The following table describes the groups:

ADH Curve Groups

Name	Description	Specification Use	Restrictions
Depth	Time offset with depth measurement (L)	Flow or pressure boundary condition (DB OVH or DB LDH card)	
Discharge	Time offset with flow rate (L^3/T)	Flow boundary condition (NB DIS card)	
Draft	Time offset with draft measurement (L)	Pressure boundary condition (DB LID card)	
Elevation	Time offset with elevation value (L)	Pressure boundary condition (DB LDE card)	
Flow (per unit length)	Time offset with unit flow rate (L^2/T)	Flow boundary condition (NB OVL card in conjunction with an ADH edge string)	
Meteorologic	Time offset with unit flow rate (L/T)	Flow boundary condition (NB OVL card in conjunction with an ADH material string)	
Time Step Size	Time offset with time step size (T)	Simulation condition (TC IDT card)	Unavailable in steady state
Transport	Time offset with concentration measurement (PP)	Transport boundary condition (NB TRN or DB TRN card)	Unavailable in steady state
Velocity	Time offset with velocity components (L/T) (three columns)	Flow boundary condition (DB OVH or DB OVL card)	
Water Surface Elevation	Time offset with elevation value (L)	Flow boundary condition (NB OTW card)	
	(Units: T = time, L = length, PP = unitless parts per)		

Curves with similar value types (i.e. pressure boundary conditions) have been separated into individual groups to avoid the unintentional application.

Specifying and Selecting Time Series Data

In the ADH menu, selecting the Time Series Data... option will open the Time Series editor with the above mentioned available curve groups. This is provided so that the user can quickly view or specify all simulation time series data from one window. However, specifying a time series data in the editor accessed from this menu item does not associate the time series with a boundary or simulation condition. Various windows of the ADH interface (i.e. Boundary Condition tab of the Material Properties window) include a curve button which opens the Time Series editor with only the condition's curve group available. Association of the selected curve with a condition occurs when the editor's OK button is selected. These buttons will display the name and plot of the selected time series, unless in a spreadsheet (i.e. Transport Constituents tab of the Boundary Condition Assignment window).

Steady State Simulation

If setting up a steady state simulation, the Time Series editor is still used to specify and associate time series. There is no restriction on the time series data in the interface (the number of rows is not limited), however SMS will write a steady state series consisting only of the first condition value(s) listed. The written series is created using the simulation start and end times (zero offset and the simulation length from the Time tab in Model Control, respectively) with the same condition value(s). To avoid confusion, it is suggested that only one row should be specified in the editor (with a zero time offset) for all steady state series. SMS will validate that a time series selected to be associated with a boundary condition is appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), only the time series that are actually used by the simulation (associated with conditions) are written out. These are converted to the units expected by ADH (i.e. mph to ft/s) as necessary and written as XY1 cards with XYT cards if curve fit tolerance is specified (apart of time series attributes). Unused curves can be saved by exporting them from the Time Series editor. A written series that is not defined for the full simulation duration is automatically appended with the simulation end time and the last specified condition value(s). A time series in the file can appear different than as specified in the Time Series editor because of unit conversions and validation.

After reading a XY1 card from a file, SMS will assign the data to a curve group based on the type of card(s) that references it in the file. If multiple types of cards reference the same time series, then the data will be assigned to multiple curve group as separate and unique time series.

Go to SMS:ADH page

ADH Time Series Attributes

This window is accessed from the Time Series window by clicking on the Attributes... button. The current curve group must be an ADH model specific curve group (i.e. ADH Discharge) to open this attributes window. The attributes are only associated with the selected curve.

A curve fit tolerance (XYT card) can be specified with each curve to be used when ADH interpolates values from a curve (while time adapting).

Dialog Description

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance is included and its value. The edit field is enabled only if the check box is checked.

Go to SMS:ADH page

ADH Velocity Series Editor

The Velocity Series Editor is used to define velocity versus time for ADH Dirichlet flow velocity boundary conditions. A velocity must be defined by specifying its x and y components. The components are used to calculate the velocity magnitude and direction.

Dialog Controls

Data spreadsheet contains three columns for time and velocity components. Define the series by entering values into each cell. When the empty row (last row) is edited, a new empty row will be appended. The empty row does not have any value and is not part of the series definition. An undefined series consists of only the empty row. Every cell of the series (the empty row is excluded) must contain a value and cannot be "blanked out" (left undefined). Copy and paste are enabled for the cells, and the spreadsheet cells include a right-click menu.

Insert New Row Above tool and spreadsheet right-click menu option will insert a row above the spreadsheet selection. The new row's values will be interpolated from the rows directly above and below, unless the new row is inserted above the spreadsheet's first row, in which case the values will be copied from the row directly below. This is only available if there is a spreadsheet selection that is not limited to just the empty row.

Delete Row(s) tool and spreadsheet right-click menu option will delete the row(s) included in the spreadsheet selection. The entire row(s) will be removed from the spreadsheet and all rows below the deletion will be shifted up. Since every cell of the series must contain a value, to remove a specific value, that value must be edited or the entire row (all three values) must be removed.

Curve Name text edit field specifies the name that will be associated with the curve defined in the spreadsheet. The ADH Dirichlet flow velocity series are written to file as two separate curves (velocity x component versus time and velocity y component versus time, using the same time values) so this name will be applied to and written out with both series definitions.

Go to SMS:ADH page

ADH Vessel Coverage

The ADH Vessel Coverage is used to add vessels to the simulation and give them paths to follow. Generally, one coverage represents one vessel. You can drag one or more vessel coverages into the ADH Mesh to add those coverages to the simulation (by creating links). This allows you to have more vessels defined than you are using in your simulation, and you can swap them in and out of the simulation to run different tests with different vessels.

Boat Path

Each vessel coverage has at least one arc to define the boat path. The arc determines the vessel's starting position and starting speed, and where it will go from there. Additional segments can be added to the arc by adding vertices. Vertices add destinations but do not affect speed. To change the speed of the boat you can convert a vertice into a node, and enter the node's attributes.

In general, there will always be exactly one path per vessel coverage. The path can be made up of multiple arcs, but the arcs should not split into multiple paths. The one exception to the rule of "one path per coverage" is when you have multiple vessels that are identical except for their speeds and paths. In this case you can have multiple separate paths in the same coverage, and each will create its own separate copy of the boat defined in the coverage properties. These paths can cross each other as long as they do not connect to each other (at a node). Another option is to simply duplicate the coverage after the boat properties are defined. This allows you to add and remove the boats from the simulation separately, and the boats can have vertices and nodes in the same place (same x and y coord) along their path without conflicting. The boat path writes the FDEF and SDEF cards to the boat file.

Dialogs

Node Attributes

Nodes on arcs change the speed of the boat or, in the case of the first node of the path, define the boat's starting speed. To change the speed of the boat at a node, select it and right-click, then select 'Node Attributes...'. Alternately, you can double-click the node.

Boat Parameters

The boat's parameters are defined in the coverage properties. Right-click on the coverage and choose 'Properties...'. This dialog lets you set the boat's size and shape, and define propellers if desired. If the OP BTS card is included, each vessel in the simulation will need to have propellers defined. Without the OP BTS card, propellers supposedly do nothing.

Each field corresponds directly to a card in the boat file.

- Length (BLEN)
- Width (BWID)
- Bow to Length Ratio (PBOW)
- Stern to Length Ratio (PSTR)
- Draft (DRFT)
- Fraction of Draft Applied to PBOW (CBOW)
- Fraction of Draft Applied to PSTR (CSTR)

Propeller (PROP card):

- Propeller Type: Open wheel or Kort nozzle
- Propeller Diameter

- Distance between propellers
- Tow boat length: This length provides an offset distance of the propeller induced shear stresses from the vessel. Set to 0 if there is no tow boat.
- Distance from prop to tow boat stern

Go to SMS:ADH page

ADH Wind Stations

Wind station data can be included in an ADH simulation to apply wind stress on the water surface. This data is in addition to and should not affect the mesh geometry, so stations are specified on a feature coverage in the Map module. The Spatial Data coverage is used because it allows the specification of generic time series data at locations represented by feature points. ADH wind station data is specified in SMS as wind speed data (NOTE: ADH currently expects wind stress data, but is planned to accept wind speed data.)

Creating Wind Stations

To create a wind station for ADH, first create a new Spatial Data coverage and give it a descriptive name. Using the Create Feature Point tool, add a point at the location of the wind station. Right click on the feature point with the Select Feature Point or Node tool, point at the Add option, and select Time Series. This will open the Time Series editor. Select a velocity curve group (either "Velocity – Components" or "Velocity – Mag. & Dir.") and click on the New... button. Specify a name, curve data, and reference time. When finished, clicking the OK button adds the selected curve to the selected feature point's data. Right clicking on the feature point again, this menu will display the added time series and provide options to view, edit, or delete this data.

After wind stations have been created on a coverage, the ADH simulation must be told to include this information on the Model Parameters page of Model Control. Since a specific coverage is choosen, the user can create multiple coverages and quickly switch wind conditions as necessary. The velocity time series data of a coverage is not specific to ADH, and consequently do not include ADH time series attributes which contains curve fit tolerance. Therefore, a general curve fit tolerance can be specified in Model Parameters to be attributed to every velocity time series in the selected coverage.

Steady State Simulation

If setting up a steady state ADH simulation, a spatial data coverage is still used to specify wind stations and their time series. However, SMS will write a steady state velocity series consisting only of the first condition values listed regardless of the reference time. The written series is created using the simulation start and end times (zero offset and the simulation length from the Time tab in Model Control, respectively) with the same condition values. To avoid confusion, it is suggested that only one row should be specified in the editor (zero offset value with any reference time) for all steady state velocity series. The user is responsible for specifying appropriate steady state data.

Boundary Condition File

When saving the ADH Boundary condition file (*.bc), SMS will check to whether wind station data is included and, if specified, that the selected coverage is still valid (exists). SMS will then proceed to iterate through every feature point in the selected coverage, formatting series data and writing XYC and XY2 cards (with XYT cards if curve fit tolerance is specified). Since the general velocity types (Velocity – Components and Velocity – Mag. & Dir. curve groups) include reference time, part of the series formatting includes extracting data and re-referencing it to the simulation start time. If the data does not cover the simulation start or end time, it will be appended with the time and the first or last specified condition values, respectively. The extracted series will start at the simulation start time and include all remaining data. Formatting also includes converting velocity data to component form (magnitude and direction to x,y components) and units expected by ADH (i.e. mph to ft/s) as necessary. A velocity time series in the file can appear different than as specified in the Time Series editor because of form and unit conversions, and validation. Any feature point without velocity data will be ignored along with all other feature objects the coverage may contain. If a feature point contains multiple velocity time series, then only the data of first time series found will be written. Coverages not included in the simulation can be saved separately if desired.

As SMS reads an ADH boundary condition file, wind station data will be added to a new spatial data coverage with the default name of "ADH Wind Station". This coverage will be automatically selected in Model Paramters. If the file contains any curve fit tolerances for wind station velocity time series, the minimum tolerance (the values may be different) will be used for the entire coverage.

Go to SMS:ADH page

6.1.a. ADH Boundary Condition

ADH Boundary Condition

To open this window, at least one node or nodestring must be selected. After a selection is made, select the *ADH | Assign BC...* menu item or use the right-click menu.

ADH boundary conditions consist of two specification types (Dirichlet and Natural) and three condition types (flow, pressure, and transport). Dirichlet data is applied on the domain to individual nodes or to groups of nodes (defined with nodestrings). Natural data (flux) is applied through edges of the domain defined by nodestrings and includes a friction specification. A node or nodestring can be assigned only one condition option from each of the three condition types (two flow conditions cannot be assigned to a single node).

Boundary Conditions

Flow Condition Options	Specification Type	Information
None	---	No condition specified
Flow (per unit length)	Natural	Specified by a Flow (per unit length) time series; NB OVL card with a friction card
Sidewall (no through flow)	Natural	Creates a vertical wall with the specified shear stress; just a friction card
Subcritical inflow	Dirichlet	Specified by a Velocity time series; DB OVL card
Supercritical inflow	Dirichlet	Specified by a Depth time series and a Velocity time series; DB OVH card
Tailwater (outflow)	Natural	Specified by a Water Surface Elevation time series; NB OTW card with a friction card
Total discharge	Natural	Specified by a Discharge time series; NB DIS card with a friction card

Pressure Condition Options	Specification Type	Information
None	---	No condition specified
Lid (depth underneath)	Dirichlet	Specified by a Depth time series; i.e. a structure (ex. culvert, sluice gate, etc.) defined using bed elevation as datum; DB LDH card
Lid (draft)	Dirichlet	Specified by a Draft time series; i.e. a floating object (ex. boat, dock, etc.) with draft; DB LID card
Lid (elevation)	Dirichlet	Specified an Elevation time series; i.e. a structure (ex. culvert, sluice gate, etc.) defined using the elevation datum; DB LDE card

Transport Condition Options	Specification Type	Information
None	---	No condition specified
Dirichlet	Dirichlet	Specified by a Transport time series; DB TRN card
Equilibrium	Dirichlet	Specified by initial concentration (in ppm); This condition can only be assigned to sediments and specified in simulations without cohesive sediments; EQ TRN card
Natural	Natural	Specified a Transport time series; NB TRN card

Weir Options	Specification Type	Information
None	---	No condition specified
Weir		Nodestrings are specified as upstream, upstream edge, downstream or downstream edge. See the WER and WRS card

Natural Flow Condition Friction Options

Friction Options	Information
Manning's n	Specified by a roughness value; FR MNG card
Equivalent roughness height	Specified by a roughness height (in ft or m); FR ERH card
Submerged aquatic vegetation	Specified by an undeflected stem height (in ft or m); FR SAV card
Un-submerged rigid vegetation	Specified by a roughness height (in ft or m), averaged stem diameter (in ft or m), and average stem density (in stems/ft² or stems/m²); FR URV card

All mesh boundary edges that do not have a boundary condition will be treated as a frictionless sidewall with no through flow.

All boundary conditions listed above can only be assigned to the mesh boundary, expect the pressure conditions which can also be assigned to the mesh interior.

When changing a condition type on any tab, the existing condition information is cleared from the window such that the information must be respecified upon returning to that type.

Dialog Description

The following controls specify the boundary condition to be assigned to the currently selected entities. This window is expandable by dragging the window edges to increase visibility of the spreadsheet data on the *Transport tab*.

Boundary Condition Type (Tab Control)

This tab control can contain the following three tabs:

Flow

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary.

Boundary type combo box contains the conditions available for flow (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will open the Time Series editor (limited to the curve group of the boundary type). A second curve button will appear for boundary types that require two conditions with time series data.

Friction combo box specifies the type of bed shear stress to be applied to the boundary edge and is available only for Natural boundary types. The friction options are:

- *Manning's n* provides the *Manning's n* roughness real number edit field limited to a value of 0.0 to 0.3.
- *Equivalent roughness height* provides the *Roughness height* positive real number edit field.
- *Submerged aquatic vegetation* provides the *Undeflected stem height* positive real number edit field.
- *Un-submerged rigid vegetation* provides the *Roughness height, Average stem diameter* and *Average stem* density positive real number edit fields.

Pressure

This tab is always available.

Boundary type combo box contains the conditions available for pressure (listed above) for the selected entities.

Series data curve button selects the time series to be associated with the selected boundary type. The text label above this curve button will change with the boundary type to denote the type of input. Clicking on the curve button will open the Time Series editor (limited to the curve group of the boundary type).

Transport Constituents

This tab is only available if all of the currently selected nodes or nodestrings are on the mesh boundary and constituents have been included in Model Control Transport Constituents.

Boundary type combo box contains the conditions available for transport (listed above) for the selected entities.

Spreadsheet contains the following information specific to each transport type:

- *None* clears the display of any information in the spreadsheet.
- *Dirichlet* provides each defined constituent (first column) with a *Include* check box (second column), *Specify Series...* curve button (third column), and a series name cell (fourth column). To include the constituent in the specification of the transport condition, check the *Include* control and specify a concentration series by clicking on the *Specify Series...* button which will open the Time Series editor (limited to the Transport curve group). When a series has been defined, the name will be listed in the series name cell. A series can be selected by multiple constituents. The concentration series button and name cells are only enabled if the constituent is included in the transport source.
- *Equilibrium* provides each defined sediment (first column) with a Include check box (second column) and a positive real number *Initial concentration* cell (third column). To include the sediment in the specification of the transport condition, check the *Include* control and specify a concentration (in parts per million). If the sediment is not to be present at the beginning of the simulation, specify a concentration of zero. The *Initial concentration* cell is only enabled if the sediment is included in the transport source.
- *Natural* provides the same as *Dirichlet.*

Output (untitled group box)

Output calculated flow across this entity check box is available only for nodestrings and correlates to the FLX card.

Go to SMS:ADH page

ADH Boundary Condition File Cards

Go to SMS:ADH page

The following lists ADH *.bc cards that are (or will be) recognized by SMS and their capability (reading, editing values and writing) currently available within the interface. For a description of card specifics, please use the ADH Quick Reference [1]. SMS will ignore any card that is not listed below when reading a *.bc file.

Constituent Properties

See Model Control Transport Constituents for interface information.

		Read	Edit	Write	Notes
CN	CLA	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	CON	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SAL				
CN	SLT	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	SND	X	X	X	SMS writes a constituent name character string in quotes preceding the regular card fields.
CN	TMP				
CN	VOR	X	X	X	

Top of page

Friction Controls

See Boundary Condition Assignment for interface information.

		Read	Edit	Write	Notes
FR	ERH				
FR	MNG	X	X	X	A natural boundary condition without a specified friction card will have a roughness value of zero.
FR	SAV				
FR	URV				

Top of page

Iteration Parameters

See Model Control Model Parameters for interface information.

	Read	Edit	Write	Notes
IP FLI	X	X	X	This card and the IP MIT card are mutually exclusive.
IP FNI	X	X	X	This card and the IP NIT card are mutually exclusive.
IP ITL	X	X	X	
IP MIT	X	X	X	This card and the IP FLI card are mutually exclusive.
IP NIT	X	X	X	This card and the IP FNI card are mutually exclusive.
IP NTL	X	X	X	
IP RTL				
IP SST				

Top of page

Material Properties

See Model Control Global Material Parameters and Material Properties for interface information.

	Read	Edit	Write	Notes
MP CBA				
MP COR	X	X	X	Material specific.
MP DF				Material and transport constituent specific.
MP DTL	X	X	X	Global property.
MP EEV	X	X	X	Material specific.
MP EVS	X	X	X	Material specific.
MP G	X	X	X	Global property.
MP ML	X	X	X	Material specific.
MP MU	X	X	X	Global property.
MP MUC	X	X	X	The ADH units system flag currently keys off of the current horizontal coordinate system.
MP NBL				
MP RHO	X	X	X	Global property.
MP SBA				
MP SBM				
MP SBN				
MP SRT	X	X	X	Material specific.
MP TRT				Material and transport constituent specific.

Top of page

Operation Parameters

See Model Control Model Parameters for interface information.

		Read	Edit	Write	Notes
OP	BLK	X	X	X	
OP	BT	X	X	X	
OP	BTS				
OP	INC	X	X	X	
OP	PRE	X	X	X	
OP	SW2	X	---	X	Always written by SMS (not editable), denotes a 2D Shallow Water ADH model
OP	TPG				
OP	TRN				
OP	TEM				

Top of page

Output Controls

See Model Control Output for interface information.

	Read	Edit	Write	Notes
END	X	---	X	Always written by SMS (not editable), denotes the end of the file.
FLX	X	X	X	See Boundary Condition Assignment for additional interface information.
OC	X	X	X	

Top of page

Solution Controls

See Boundary Condition Assignment for nodal and nodestring boundary condition interface information.

		Read	Edit	Write	Notes
DB	LDE	X	X	X	
DB	LDH	X	X	X	
DB	LID	X	X	X	
DB	OVH	X	X	X	
DB	OVL	X	X	X	
DB	TRN				
EQ	TRN				
NB	DIS	X	X	X	
NB	OTW	X	X	X	
NB	OVL	X	X	X	See the Boundary Condition Assignment section of Material Properties for additional interface information.
NB	TRN				
OB	OF	X	---	---	This card is obsolete as the functionality is now handled internally in ADH.

OFF	X	X	X	This card can only be applied to a material string (MTS). See the Properties section of Material Properties for interface information.

Top of page

String Structures

See Practical Notes for string structure information.

	Read	Edit	Write	Notes
EGS	X	X	X	Use the Select Nodestring tool, and assign boundary conditions.
FCS				
MDS	X	X	X	Use the Select Nodestring tool, and assign boundary conditions for flow output.
MTS	X	X	X	Add and delete materials in the Materials Data window. Specify material properties, and using the Select Element tool, assign material type from the Elements | Assign Material Type... menu command.
NDS	X	X	X	Use the Select Node or Select Nodestring tool, and assign boundary conditions.

Top of page

Time Controls

See Model Control Time for interface information.

		Read	Edit	Write	Notes
TC	IAC	X	X	X	
TC	IDT	X	X	X	
TC	NDP	X	X	X	
TC	SDI	X	X	X	
TC	STD	X	X	X	
TC	STH				
TC	TF	X	X	X	
TC	T0	X	X	X	SMS writes a "global time" character string in quotes preceding the regular card fields. The date/time information is used to relate the simulation with other data loaded in SMS concurrently.

Top of page

Time Series

	Read	Edit	Write	Notes
XY1	X	X	X	Define series (curves) in various places. An output times series using the auto-build specifier can be read in and created, however, the auto-build format is not currently used when writing out. SMS writes a series name character string in quotes preceding the regular card fields.
XYC				
XYT	X	X	X	Available where series (curves) are defined as "curve fit tolerance".

Top of page

Go to SMS:ADH page

6.1.b. ADH Model Control

ADH Model Control

Go to SMS:ADH page

The AdH Model Control dialog contains the graphical prompts for specifying model options. The dialog is accessed through the ADH | Model Control... menu item.

Dialog Description

Model control contains the following five pages:

- Model Parameters
- Iterations
- Time
- Output
- Global Material Properties
- Transport Constituents
- Conslidation
- Advanced (Cards)
- Solver

ADH Model Control Output

This page window is accessed from the Model Control window by clicking on the *Output* tab.

ADH will write solution data at startup and at each time step specified in a XY1 series definition and referred to by the OC card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

ADH Model Control Advanced (Cards)

Model options not available in the interface can be specified in the ADH model control under the advanced tab. Each row in the spreadsheet will be saved as a different line in the *.bc file. The cards can be inserted and deleted using the insert and delete rows.

- The lines will be written out exactly as typed.
- The lines will be written out in the order in which they appear in the dialog.
- Any lines encountered in a *.bc file that is not recognized will appear in this list.
- Lines beginning with a "!" character are treated as comment lines and will be skipped over during a read.
- Comment lines are automatically generated and will not be preserved.
- All of these advanced cards will be grouped together at the bottom of the file regardless of where they appeared during reading.

Dialog Description

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes Start at positive integer/real number field and units combo box, End at positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End* at time units are the same as specified for the *Start* at time. The Add individual output time control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc., with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Go to SMS:ADH page

ADH Model Control Model Parameters

This window is part of the AdH model control dialog (AdH|Model Control... command). It is accessed from the Model Control window by clicking on the Model Parameters tab.

Model Parameters

The model parameters window is divided into four sections including:

- Operation
- Wind Stations
- Density effects
- Bendway correction

Each of these sections are described below.

Operation

The operation section contains controls to specify general operation parameters for AdH. SMS automatically writes the OP SW2 card to all AdH boundary condition files because SMS interfaces with the 2D shallow water problems simulated by AdH. The other operational parameters specified in this section include:

Memory allocation block size (OP INC card)

The *Incremental memory allocation* block size edit field allows the user to specify a positive integer defining the size of the incremental memory allocations. The default value is 40 units and no explanation has been provided for when or what to change this to. We recommend most users leave the default value.

Iterative solver pre-conditioner (OP PRE and BLK cards)

The *Iterative solver pre-conditioner* parameters apply to parallel processing applications. If you are running AdH in single processor mode, this card has no impact.

If you are running AdH in parallel mode, the edit field defines how many blocks per processor are to be used in the preconditioning which subdivides the problem for each processor. No guidance has been provided by the AdH development team about when to use different pre-conditioning schemes. We recommend the defaults of "One-level Additive Schwarz" and "1 block per processor".

Vessel stress effects (OP BT card)

This toggle instructs SMS to write the OP BTS card which includes vessel stress effects check in the simulation when checked. This calculates and outputs bed shear stresses due to vessels in dyn/cms. This currently requires use of metric units. If you enable this option, make sure you have vessel coverages in your simulation and that each boat has propellers defined.

Wind Stations (XYC cards)

Wind can be included in a number of ways in an AdH simulation. These include on a material by material basis described in a different section. The *Include wind stations (defined on spatial data coverage)* check box lets the user specify that wind shear stress should be included in the simulation, computed from a set of spatially defined wind stations. The Include check box is only available if at least one spatial data coverage exits. In the spatial data coverage the user should define a set of wind gages (defined as feature points) with time series of wind data associated.

If more than one spatial data coverage is defined, the associated *Select...* button undims and the user can select a single set of wind data for the current simulation. Clicking on the *Select...* button opens the *Select Coverage* window which lists all available spatial data coverages.

SMS will write an XYC card for each wind station. This card defines the curve containing the wind data and the location of the station.

Specify curve fit tolerance check box and the associated positive integer edit field specify whether a tolerance for all wind time series is included and its value. These controls are enabled only if a coverage is selected (the tolerance check box must also checked to enable the edit field).

For more information see Wind Stations.

Density effects

The AdH model includes the ability to simulate the effects of salinity and temperature on flow. This section includes two toggles for the user to activate these options.

Salinity (CN SAL card)

The *Include salinity* check box with the associated Reference concentration allow the user to turn on the simulation of salinity and define a positive non-zero real number for the background or reference concentration . The concentration is specified in parts per thousand.

Temperature (CN TMP card)

The *Include temperature* check box with the associated *Reference temperature* allow the user to turn on the inclusion of temperature effects in the simulation of specify a positive non-zero real number which correlates to the CN TMP card reference temperature. The temperature is measured in degrees.

Bendway Correction (CN VOR card)

The SW2 option in AdH simulates two-dimensional flow. However, some three dimensional effects can be approximated. This includes a method for correcting for the effects of vorticity around bends. Selecting the *Include vorticity* check box will cause SMS to add this card and activate this feature. It also adds the OP TRN card since vorticity is simulated in AdH as a transport constituent.

When the feature is active, The user must also specify three coefficients. These include the As and Ds empirical coefficients determined by integrating against measured values (defaults of 5.0 and 0.5) and a normalization factor.

The *Override empirical coefficients* toggle activates the edit fields for the other coefficients. If the option is unchecked, the coefficients will be reset to the default values. Zero values for both coefficients will assume the default values. Currently we do not recommended overriding the empirical coefficients.

Go to ADH Model Control page

ADH Model Control Iterations

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Iterations* tab.

Iterations

The iterations tab allows the user to specify the level of precision for the conservation of mass and momentum as ADH performs its calculations.

ADH solves conservation statements concerning water volume, momentum, and constituent mass. ADH is written in conservative form and thus can be regarded as both a finite element method and also a finite volume method. As such ADH can be shown to represent a sum of fluxes around the edge of an element to be in balance with the mass or volume change within the element. In the case of the momentum equation it will be a sum of fluxes and forces balancing the momentum change within an element.

ADH computes the water levels and velocities at each computation point in the domain for each time step in the simulation in an iterative process. At the end of each iteration, calculations are made to determine how much the solution is changing and what the errors in conservation are. If the errors are small, and the solution is not changing, the process is said to be converged and calculations can proceed to the next time step. If the solution is changing and/or the errors in conservation are large, ADH can attempt another iteration.

There are several iteration parameter controls in ADH. These are described in detail in the ADH users manual [1]. The Solver options tab controls the linear iterations. This dialog controls the non-linear iteration parameters.

At the end of each iteration ADH computes the conservation properties for the current approximations of water depth and velocity. If the errors are less than specified tolerance, the solver moves on to the next time step. If the errors are greater than the specified tolerance, but still reasonable, and the maximum number of iterations specified on the IP NIT card has not been reached, ADH will attempt another iteration. If the errors have become unreasonable (divergence), ADH will exit the loop with a failure notification. The controls in this dialog allow the user to define how ADH should determine convergence and how to proceed when convergence is not reached.

Maximum number of iterations per time step (IP NIT and IP FNI cards)

In this edit field the user specifies the maximum number of iterations that ADH will try for a single time step. By examining the output for successive iterations, a user can determine if the process is converging in general (trending towards zero), or oscillating. If the trend is towards convergence, this maximum number of iterations can be increased. As the value increases, ADH can spend more time processing individual time steps which may decrease the overall efficiency of the solver. For this reason, a user should not simply increase this parameter without verifying that it will be beneficial.

SMS writes either the IP NIT or IP FNI card based on the setting of the If tolerance is not satisfied radio group.

- IP NIT card is written if the Reduce timestep option is selected. (Recommended)
- IP FNI card is written if the Accept solution option is selected.

The IP FNI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the Reduce timestep option is selected and the solution has diverged or the maximum number of iterations have been attempted, ADH will reduce the time step size to ¼ of the previous and continue the calculations.

When the Accept solution option is selected, and the maximum number of iterations have been attempted, ADH proceeds as if convergence has been reached to the next time step. (This option is not recommended.)

NTL/ITL/Both combo box

NTL (IP NTL card)

If the NTL option is selected, ADH will check for non-linear convergence on the "maximum residual norm". An initial estimate for an appropriate tolerance value for a problem can be computed as:

$\frac{(total_area * 10^{-6})}{\#_elements}$This is the maximum allowance, so you will likely want to set the tolerance higher than this estimate so your simulation will progress.

ITL (IP ITL card)

If the ITL option is selected, ADH will check for non-linear convergence on the "maximum increment norm" or the change in the solution (velocity, depth and concentration).

Both

If the Both option is selected, SMS will write both cards to the BC file and ADH will check for convergence on both terms. Both must be satisfied for the solution to proceed.

Go to SMS:ADH page

References

[1] http://chl.erdc.usace.army.mil/chl.aspx?p=m&a=MEDIA;1225

ADH Model Control Time

Go to ADH Model Control page

This page window is accessed from the Model Control window by clicking on the *Time* tab.

ADH is set up as a transient conditions model, however, a steady state simulation can be invoked which utilizes an iterative process to converge to a solution. A steady state solution is accepted if the end time of the model is reached or the iteration tolerance requirement (see Model Parameters) is met. Users are required to ensure that all series data are acceptable (constant linear series) if the steady state method is used. ADH also includes a quasi-unsteady method which strings multiple steady state simulation together to form a step function style hydrograph.

Due to the nature of a steady state simulation, constituent transport and bed layers cannot be included in the simulation. If any constituent (and bed layer) is specified in a dynamic simulation and then the simulation is changed to steady state, SMS will clear the constituent data (and bed layer specification). A message will warn the user of this effect when the steady state simulation type is selected and constituent data (and a bed layer) exists. Another message will state that SMS has cleared the data (if steady state is still selected) when switching to another tab or exiting Model Control by selecting OK.

Since series data specified for a dynamic simulation is usually inappropriate for a steady state simulation, a change in simulation type from dynamic to steady state will force SMS to clear all previously specified series data when changes are accepted in Model Control. This data is usually associated with the boundary conditions of materials, nodes, and nodestrings. A message will warn the user of this effect when the simulation type is changed and series data exists. Another message will state that SMS has cleared the data (if the simulation type remains changed) upon accepting the changes (exiting Model Control).

Time

The following controls specify the timing parameters for the ADH model run.

Simulation

Dynamic / Steady state / Quasi-unsteady radio group specifies the simulation type.

- *Dynamic* option provides the following:
  - *Start* date and time field and the *Delay model start* check box with positive real number and units combo box correlates to the TC T0 card.
  - *End* date and time field (with the defined start) specifies the simulation run length on the TC TF card.
  - *Duration* positive real number edit field and units combo box also specifies the simulation run length. This field and the End field will update as the other is changed.
- *Steady state* option provides the following:
  - *Duration* positive real number edit field and units combo box to specify the simulation length.
- *Quasi-unsteady* option is currently unavailable.

Apply adaptive time control check box includes the TC IAC card or TC NDP card when checked or unchecked, respectively.

Time Step Control

Time step size curve button with the associated *Series time units* combo box, and *Specify curve fit tolerance* check box and positive integer edit field defines the data to be associated with the TC IDT card and its XY1 series definition card. Clicking on the curve button will open the XY Series Editor. This series must be defined for the entire model run length, so include or exceed the time *Model run time* listed in the *Simulation* group. This control group is available only when the *Dynamic* simulation type is selected.

Initial time step positive real number edit field and units combo box specifies the time step value of the TC STD card. This control group is available only when the *Steady state* simulation type is selected.

Specify extra sediment transport time steps check box and positive integer edit field while include the TC SDI card when checked. If sediment transport is included in the simulation and this control is not checked, then the model will calculate sediment according to the time increment specified by the *Time step size* series.

Dynamic simulation type option, in the Simulation group, provides the following:

- *Time step* size curve button selects the time series data to be associated with the TC IDT card. Clicking on the curve button will open the Time Series editor (limited to the Time Step Size curve group). The selected series should be defined for the entire model run length, so include or exceed the time Model run time listed in the Simulation group.
- *SNumber* of sediment transport time steps per hydrodynamic time step positive non-zero integer edit field correlates to the TC SDI card. This field is only enabled if sediments have been specified on the Transport Constituents tab of Model Control. This applies only to sediments and not any other constituents. If sediment transport is not included in the simulation, then the TC SDI card is not included in the file.
- *Max sediment time steps* value is associated with the TC SDI card. This allows users to specify smaller sediment transport time steps that what is used for the hydrodynamic time step. Sediment constituents must exists otherwise this option will be grayed out and the card will not be written.
- *Automatic time step* option, provides the following:
- Specifies the time step data for the TC ATF card. Pressing the 'calculate' button calculates the time step based on the formula: $= x / \sqrt{g*h}$

  Where:

x = mesh element length (currently the distance between the 1st and 2nd node in the 1st element)

g = gravitational acceleration

h = min water depth

If an initial dataset doesn't exist, the automatic time step option is disabled and the TC ATF card is not written.

Steady State simulation type option, in the Simulation group, provides the following:

- *Initial time step* positive real number edit field and units combo box specifies the time step value of the TC STD card. This control group is available only when the Steady state simulation type is selected.

Go to ADH Model Control page

ADH Model Control Output

Go to ADH Model Control page

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Output* tab

ADH will write solution data at startup and at each time step specified in a XY1 series definition and referred to by the OC card. Flux data will be included with the solution data if output flow strings are specified (see assigning boundary conditions).

Output Control

The following controls specify how often solution data will be written by ADH and the number of flux boundaries present.

Output Times

List box contains the times to be contained in the output series. Multiple selection is enabled for deleting times.

Add by specifying a range / Add individual output time radio group specifies time information to be added. The *Add by specifying a range* control group includes *Start at* positive integer/real number field and units combo box, *End at* positive integer/real number field, and *Increment* positive integer/real number field and units combo box. The *End at* time units are the same as specified for the *Start at* time. The *Add individual output time* control includes a positive integer/real number field and units combo box. Each edit field allows for positive integer seconds or positive real number minutes, hours, etc with six decimal places. All times are rounded to the nearest whole second (a decimal minute is rounded to the nearest decimal minute with a whole second equivalent) before being added.

Add button includes the time information from the above radio group in the times list box. Duplicate times will be automatically removed from the list.

Delete button removes the selected time(s) from the list box.

Delete All buttons clears the entire times list without regard to the current selection.

View output times in combo box specifies the time unit the list box will display the output times in.

Specify curve fit tolerance check box and positive integer edit field specifies the series tolerance when checked.

Flow

Number of nodestrings specified to output flow calculated across themselves displays the total number of ADH flux boundaries included in the simulation.

Geometry

Output adapted mesh files check box specifies whether geometry files will be saved as ADH adapts the mesh. The geometry will be outputted at the same interval specified in the list box for the solution data. This correlates to the PC ADP card.

Go to ADH Model Control page

ADH Model Control Global Material Properties

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Global Material Properties* tab.

ADH contains parameters that apply to all materials and parameters that are specific to (and must be defined for) each material. The parameters found within this tab are of the former type.

The bed layers specified here are applied to all mesh nodes and can only be overwritten at the nodal level in the Bed Layers Assignment window. When bed layers are created or deleted here, nodal bed layers are defaulted or deleted, respectively.

Global Material Properties

Parameters (Untitled group box)

Enable wetting/drying limits check box and the associated *Dry depth* and *Wet depth* positive real number edit fields correlate to the optional MP DTL card. If the water depth of a node is below the *Dry depth*, then the node is completely dry and is not included in the ADH shallow water equations. If the water depth is above the wet depth, then the node is included in the equations. A water depth which falls within these two depths will include the node in the equation with a restricting factor.

Constant

Uniform background viscosity positive real number edit field correlates to the MP MU card.

Gravitation acceleration positive real number edit field correlates to the MP G card.

Density positive real number edit field correlates to the MP RHO card.

Wetting and Drying

Enable shock capturing/stability procedures below depth is check box and the associated wetting and drying process. When this toggle is selected, ADH performs extra calculations for all cells with depth values less than the specified minimum to stabilize the process. As the specified depth value increases, extra calculations also increase and model performance decreases. When this option is selected, the *Depth* edit field is enabled by SMS. In this field the users specifies a positive real number correlating to the optional MP DTL card.

Sediment Bed layers

Number of bed layers positive integer edit field correlates to the MP NBL card. The value specified is the number of bed layer column provided in the spreadsheet. If the value is increased, the new bed layer columns are appended; if decreased, the existing bed layer columns are removed. The effected bed layers are always the highest numbered layers (at the end of the spreadsheet). This is enabled only if sediments have been defined on the Transport Constituents tab.

Layer 1 is the deepest (or bottom) layer; the highest numbered layer is exposed to the stream flow. The layer IDs cannot be edited and bed layering order is fixed to ascending layer ID order (See Sediment Transport and Bed Layers).

Edit spreadsheet column toolbar is enabled when a cell selection exists in the spreadsheet, and provides:

- Insert New Column Before tool inserts a single new bed layer column before (to the left of) the current selection.
- Delete Column(s) tool removes all of the bed layer columns of the currently selected cells.

Spreadsheet contains the defined bed layers as columns and bed layer specifications as rows. The bed layers begin with Layer 1 and proceeds by ascending order by ID to the right. This spreadsheet will be empty if no sediments have been defined on the Transport Constituents tab or zero bed layers are specified. The specification rows are as follows:

- Sediment distribution – Title cell with each defined sediment listed below in its own row. The row contains Normalize buttons for each bed layer. Clicking on the button will normalize the distribution of each sediment such that the total distribution equals 100%.
- Sediment (name) – Consists of this sediment's distribution (as a percentage) for each layer. A sediment row is provided for all defined sediments. The value must be a positive real number and correlates to the MP SBA card.
- Total – Displays the current total specified distribution of all sediments for each layer. The row updates automatically as specifications change and cannot be edited. Each layer's total sediment distribution must equal 100%.
- Thickness – Consists of each bed layer's thickness measured in meters. The value must be a positive real number and correlates to the MP SBA card.
- Bulk Density – Consists of each bed layer's bulk density measured in kilograms per cubic meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion crit. shear – Consists of each bed layer's critical shear stress for erosion measured in newtons per square meter. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion rate constant – Consists of each bed layer's unitless erosion rate constant. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.
- Erosion rate exponent – Consists of each bed layer's unitless erosion rate exponent. This row only appears if a cohesive sediment (clay or silt) is defined. The value must be a positive non-zero real number and correlates to the MP CBA card.

Go to ADH Model Control page

ADH Model Control Transport Constituents

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Transport Constituents* tab.

ADH transport constituents consist of two types: regular and sediment. Transport is only available for a normal (dynamic) simulation (specified on the Time tab of Model Control).

Constituents

Transport Constituents

- General – CN CON
- Salinity – CN SAL
- Temperature – CN TMP
- Vorticity – CN VOR

Sediment Transport Constituents

- Sand or gravel – CN SND
- Clay or silt – CN CLA

Non-sediment Transport Constituents

- Salinity – CN SAL (This is a currently unavailable option)
- Temperature – CN TMP (This is a currently unavailable option)
- Vorticity – CN VOR

Dialog Description

Sediment

List box contains type and name of sediments to be included in the model simulation.

New button inserts a defaulted general sediment into the sediment list.

Delete button removes the currently selected sediment from the list.

Spreadsheet contains the parameters and values specific to the currently selected sediment in the list. Available parameters will be updated when the sediment type is changed. The sediment list box will update when the sediment name or type change.

Bendway Correction

Include vorticity check box and the associated *Normalization factor*, *As coefficient*, and *Ds coefficient* positive real number edit fields correlates to the CN VOR card.

Generic and Sediment

List box contains type and name of user-defined constituents to be included in the model simulation.

New button inserts a defaulted generic constituent into the user-defined constituent list.

Delete button removes the currently selected constituent from the list.

Spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- Constituent type – Options are: Generic, Sand or gravel sediment, and Clay or silt sediment
- Name – Within SMS, the constituent will be referenced by this name. Names must be unique.
- Characteristic concentration – The characteristic concentration of the constituent is measured in micromass per unit mass or parts per million.
- Grain diameter (sand/gravel and clay/silt types) – Measured in millimeters (this is converted to meters for ADH).
- Specific gravity (sand/gravel and clay/silt types) – Value must be greater than or equal to 1.0.
- Grain porosity (sand/gravel type only)
- Bulk density (clay/silt type only) – Measured in kilograms per cubic meters.
- Erosion critical shear (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- Erosion rate constant (clay/silt type only) – Used in the erosion flux equation.
- Deposition critical shear (clay/silt type only) – Critical shear stress for deposition is measured in Newtons per square meters.
- Settling velocity (clay/silt type only) – Measured in meters per second.

The constituent list box will update when the constituent name or type changes. If the type of a constituent is changed, the name and characteristic concentration are retained, but all other values are cleared. All value fields are restricted to positive non-zero real numbers unless stated otherwise.

Go to ADH Model Control page

ADH Model Control Consolidation

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Consolidation* tab.

Consolidation

This window consists of a table defining the properties of cohesive sediments included in an ADH simulation.

The sediment classes must be specified before entering these time-based parameters for each sediment. The check box labeled Use time-based consolidation at the bottom of the page tells ADH that consolidation should be computed.

The spreadsheet contains the parameters and values specific to the currently selected constituent in the list. Constituent parameters are:

- *Time* (sec) – Measured in seconds.
- *Bulk density* (clay/silt type only) – Measured in kilograms per cubic meters.
- *Erosion crit. shear* (clay/silt type only) – Critical shear stress for erosion is used in the erosion flux equation. Measured in Newtons per square meters.
- *Erosion rate constant* (clay/silt type only) – Used in the erosion flux equation.
- *Erosion rate exponent* (clay/silt type only) – Used in the erosion flux equation.

All value fields are restricted to positive non-zero real numbers unless stated otherwise.

Go to SMS:ADH page

ADH Model Control Advanced

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Advanced* tab.

Advanced Cards

SMS includes support for many of the features in ADH, however, this list of features is a dynamic set. The user base of ADH includes a development team that is constantly experimenting with new options.

This window allows SMS to maintain the options that are not otherwise supported by the interface. For example, when SMS reads the BC file for an ADH simulation, and encounters a card that it does not recognize, that card is stored, verbatim, to a list of "Advanced Cards". That list is displayed in this window.

Users may enter cards in this dialog to experiment with other new features in the model. As those features are added to the list of supported features, they would automatically be moved to their own location in the model control the next time SMS reads the BC file.

Unsupported or advanced cards are written at the end of the BC file, just before the END card. This relies on the attribute of ADH that does not require any order dependence in the BC file.

Go to SMS:ADH page

ADH Model Control Solver

This window is part of the ADH model control dialog (ADH|Model Control... command). It is accessed from the Model Control window by clicking on the *Solver* tab.

(As noted at the top of this dialog, this is an advanced feature and we recommend that only users working with the ADH development team use these options.)

Linear Iterations By Solver

This window includes the options to control linear iterations in ADH. The controls are similar to the controls for non-linear iterations included on the Iterations tab.

The distinction between linear and non-linear iterations are described in some detain in the ADH users manual. [1]

Maximum number of iterations (IP MIT and IP FLI cards)

In this edit field the user specifies the maximum number of linear iterations that ADH will try for each non-linear iteration.

SMS writes either the IP MIT or IP FLI card based on the setting of the *If internal linear tolerance is not satisfied* radio group.

- IP MIT card is written if the *Stop the solution* option is selected. (Recommended)
- IP FLI card is written if the *Proceed to the next non-linear iteration* option is selected.

The IP FLI card instructs ADH to continue with the solution as if it had converged. This is not recommended.

When the *Stop the solution* option is selected and the solution has not converged at the maximum number of iterations have been attempted, ADH will treat the solution as diverged and the non-linear iterations will be aborted.

Go to SMS:ADH page

6.1.c. ADH Sediment Library Control

Sediment Library Control

The AdH Sediment Library Control contains the graphical prompts for specifying model options. The dialog is accessed through the ADH | Sediment Library Control menu item

Dialog Description

- Consolidation
- Transport Constituents
- Global Materials Sed Props
- Sediment Properties
- CS2SHORE Sediment

Sediment Properties

This dialog has five options.

- **Non-cohesive sediment entrainment.** Values of Option include: Garcia-Parker, Wright-Parker or Van Rijn. When this option is selected it will write out the SP NSE card.
- **Non-cohesive bedload entrainment.** Values of Option include: Van Rijn, Meyer Peter Mueller or Meyer Peter Mueller w/Wong Parker Correction. When this option is selected it will write out the SP NBE card.
- **Non-cohesive hiding factor.** Values of Option include: Karim Holly Yang or Egiazaroff. When this option is selected it will write out the SP HID card.
- **Cohesive settling velocity.** Values of Option include: Free settling or Hwang and Mehta. When this option is selected it will write out the SP CSV card.
- **Bed shear stress due to wind wave.** Values of Option include: No applied wind-wave stress, Grand and Madsen, or Teeter. When this option is selected it will write out the SP WWS card.

C2SHORE Sediment Properties

This dialog has ten options.

- Median sediment grain size with card SP C2SD50
- Breaking efficiency with card SP C2SEB
- Bottom dissipation efficiency with card SP C2SEF
- Bed porosity with card SP C2SBP
- Sediment fall velocity with card SP C2SWF
- Sediment specific gravity with card SP C2SSG
- Suspended load parameter with card SP C2SSL
- Wave-related bed load parameter with card SP C2SBLW
- Current-related bed load parameter with card SP C2SBLC
- Wave friction factor with card SP C2SFW

6.2. FESWMNS

FESWMS

FESWMS	
Model Info	
Model type	Two-dimensional finite element surface water computer program that can compute the direction of flow and water surface elevation in a horizontal plane. Also has the ability to model hydraulic structures commonly used by hydraulic engineers.
Developer	David C. Froehlich, Ph.D., P.E.
Web site	FHWA web site [1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • FESWMS • FESWMS Steering • FESWMS Weirs

The Finite Element Surface Water Modeling System (FESWMS) consists of multiple modules used to simulate surface-water flow in a two-dimensional horizontal plane. The SMS includes an interface for the FST2DH (Flow and Sediment Transport) module of FESWMS. FESWMS is sponsored by the Federal Highway Administration. David C. Froehlich, Ph.D., P.E. originally developed FESWMS for the United States Department of Transportation Federal Highway Administration (FHWA) [2] and the United States Geological Survey (USGS) [3]. The FHWA has continued to maintain and sponsor development of subsequent versions, which continue to incorporate features specifically designed for modeling highway structures in complex hydraulic environments.

Functionality

FST2DH is the FESWMS program that performs the two-dimensional hydraulic computations in surface-water bodies. FST2DH can perform either steady state or dynamic flow modeling and provides analysis of highway crossings and structures, including bridges, culverts, weirs, roadway embankments, and drop-inlet spillways. FST2DH simulates the movement of water and non-cohesive sediments in rivers, estuaries, and coastal waters by applying the finite element method to solve steady-state or time-dependent systems of equations that describe two-dimensional depth-averaged surface-water flow and sediment transport. [4]

Using the Model / Practical Notes

- FESWMS Spindown Steering (Incremental Loading)
- The representation of the wind direction in the FESWMS Manual is incorrect. The corrected image is shown in the FESWMS Model Control dialog of the SMS interface.
- FESWMS known model issues
- Support Forum FESWMS with Sediment Transport Tips User Post [5]
- Transform your mesh so it is close to the origin (0,0). Large x, y coordinates can be problematic for FESWMS.
- Verify mesh does not have incorrect elevation values (often due to extrapolation). Try looking at the mesh in a rotated view.

General Steps to build a FESWMS Model

Various sequences of steps can be used to create and edit a two-dimensional model in SMS. A suggested methodology is provided below.

1. Import background data consisting of:
    1. Elevation data covering the area to be modeled. See Scatter Module
    2. Open and register background image(s) if desired. See Images
2. Create a FESWMS coverage and define the Conceptual Model that will be used to generate the finite element mesh. See Map Module
    1. Define the physical boundaries of the model
    2. Build feature polygons within the model domain. See Build Polygons Menu
    3. Assign mesh types to the feature polygons and adjust the density and spacing of vertices (i.e., redistribute vertices) as necessary to achieve appropriate element shapes and sizes
3. Creata an area properties coverage and assign material types. This is an optional step. The FEWSMS coverage can also be used to define material regions. See Map Module
4. Generate a finite element mesh from the Conceptual Model. See Map Module
5. Check the final quality of the finite element mesh and edit the mesh as needed. See 2D Mesh Module
    1. Designate element types (i.e., linear or quadratic; triangular or quadrilateral)
    2. Review the mesh quality
6. Specify FESWMS element attributes (i.e., material properties) such as Manning's or Chezy roughness factors and eddy viscosity to the material types. See 2D Mesh Module
7. Create nodestrings and assign model boundary conditions not included in the conceptual model. See 2D Mesh Module
8. Renumber the finite element mesh.
9. Specify initial conditions and model control parameters. See FESWMS Model Control
10. Run the model.
11. View model results. If necessary, troubleshoot error/warning messages generated during the model run and revise the model accordingly.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the FESWMS model. See FESWMS Graphical Interface for more information.

The FESWMS Graphical Interface contains tools to create and edit an FESWMS simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to FESWMS. If a mesh has already been created for a FESWMS simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to FESWMS. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the FESWMS Menu. See FESWMS Graphical Interface for more information.

External Links

- Sept. 2002 (Revised October 2003) – User's Manual for FESWMS FST2DH – Publication No. FHWA-RD-03-053 [6]
- U.S. Department of Transportation Federal Highway Administration – Hydraulics Engineering Software Page [7]
- Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. [8]

Related Topics

- FST2DH Files
- FST2DH Graphical Interface
- FST2DH Hydraulic Structures
- FST2DH Model Control Dialog
- FST2DH Spindown
- FST2DH Sediment Transport
- FST2DH known model issues

References

[1] http://www.fhwa.dot.gov/engineering/hydraulics/software/softwaredetail.cfm
[2] http://www.fhwa.dot.gov/
[3] http://www.usgs.gov/
[4] Sept. 2002 - User's Manual for FESWMS Flo2DH - Publication No. FHWA-RD-03-053
[5] http://aquaveo.invisionzone.com/index.php?showtopic=332&hl=
[6] ftp://pubftp.aquaveo.com/download/s_fst2dh.pdf
[7] http://www.fhwa.dot.gov/engineering/hydraulics/software.cfm
[8] http://contentdm.lib.byu.edu/cdm/ref/collection/ETD/id/789

FESWMS Arc Attributes Dialog

The FESWMS Feature Arc Attributes dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

- Arc Type
  - None
  - Boundary Conditions – Options button opens the FESWMS Nodestring Boundary Conditions dialog
  - Continuity Check/Flux

Related Topics

- Feature Objects Menu

FESWMS BC Nodestrings

Generic 2D Mesh Boundary conditions are generally defined on feature arcs in the conceptual model or nodestrings on the 2D mesh. Boundary conditions constrain the water surface elevation and/or flow at the model boundary.

General Boundary Options

Constant vs Dynamic

FESWMS boundary conditions can be constant or dynamic. If a constant boundary condition is used with a dynamic simulation the boundary condition does not change during the simulation. It is not possible to have a dynamic boundary condition with a steady state simulation.

Total Flow String

The nodestring that is the total node flow string identifies the flowrate that will be considered 100% of the flow in the FESWMS printed output file (*.prt). Only one nodestring may be the total flow nodestring and it will be the last one identified as the total flow nodestring.

Boundary Condition Types

Specified Flow / WSE

This type of boundary constrains the flowrate and/or water surface elevation to fixed values. Both the flowrate and water surface elevation should only be specified for supercritical inflow boundary conditions.

A flow boundary condition can be a normal boundary or a weakly reflecting boundary. A weakly reflecting boundary can be used to allow tidal variation to affect the boundary. A weakly reflecting boundary simulates a fictitious river without frictional resistance upstream of the boundary.

Generally a fixed water surface elevation boundary is used for the downstream boundary condition of the model. The water surface elevation can vary from one end of the boundary to the end by using the vary along string option.

Supercritical Outflow

For supercritical boundaries, both the flowrate and water surface elevation is defined on the inflow boundary. Even though no data is associated with supercritical outflow boundaries, these boundaries must be identified so FESWMS will allow flow through the boundary.

Computed WSE

This is used to provide a rating curve (water surface vs flow) relationship to compute the downstream boundary condition. FESWMS can compute the rating curve using the friction slope option. Strangely enough FESWMS requires the user to provide flow information to use this option.

Storm/Tide WSE

This is used to tell FESWMS that the boundary water surface elevation will be determined from the storm associated with the simulation.

Related Topics

- FESWMS Arc Attributes Dialog

FESWMS Errors

This page is intended to offer suggestions on how to troubleshoot different errors when running FESWMS FST2DH.

!!!ERROR - dS = 0. This error is caused by two boundary node strings joining together at a boundary node.

If you know of an error or how to troubleshoot it, please include it here.

FESWMS Executable Known Issues

There are several different builds of the FESWMS executable (fst2dh.exe). Unfortunately there are some known issues and likely some unknown issues as well. This page is intended to help users know which executable to use for which type of project they are working on.

FESWMS development is sponsored by the Federal Highway Administration. Aquaveo does not have access to the FESWMS source code and cannot make changes to fix the issues found in the FESWMS numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of FESWMS files.

This list may not contain all know issues, so please feel free to add to it.

Sediment Transport

Aquaveo employees have successfully run test cases using the following options:

- Equilibrium inflow
- Clear water inflow

We have been unsuccessful in our attempts to create test cases using the other options [1].

Version 3.3.2

This is the version of fst2dh.exe that is currently distributed with SMS 10.0

- Does not work with Piers.

Version 3.3.2 can be downloaded at ftp://pubftp.aquaveo.com/download/fst2dh332.exe [2].

Version 3.3.3

- Fixed the issue with Piers but does not work with transient models. It is unknown what is wrong specifically other than models that run using 3.3.2 do not always work with 3.3.3. The problem seems to be that the model doesn't work with transient models.

Version 3.3.3 can be downloaded at ftp://pubftp.aquaveo.com/download/fst2dh333.exe [3].

Related Topics

- FESWMS

References

[1] Ipson, Mark K. (2006). Analysis of the Sediment Transport Capabilities of FESWMS FST2DH. Thesis, Brigham Young University. (http://contentdm.lib.byu.edu/ETD/image/etd1542.pdf)
[2] ftp://pubftp.aquaveo.com/download/fst2dh332.exe
[3] ftp://pubftp.aquaveo.com/download/fst2dh333.exe

FESWMS Files

Input and Output files for FESWMS.

Input Files

- FST2DH project file (*.fpr)
- Control data (*.dat)
- Mesh data (*.msh – *.net for SMS)
- Flow input data (*.flo)
- Sediment input data (*.sed – *.sdi for SMS)
- Boundary condition data (*.bcs)
- Wind data (*.wnd)
- Wave data (*.wve)
- Time-dependent data (*.tim)

Output Files

- Report data (*.rpt – *.prt for SMS)
- Flow output data (*.flo)
- Sediment output data (*.sed)
- Restart-recovery data (*.rsr)
- Upper coefficient matrix (*.upp)
- Lower coefficient matrix (*.low)
- Scalar output data (*.scl)
- Vector output data (*.vec)
- Profile output data (*.pro)
- Run status data (*.sta)

Related Topics

- FESWMS
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown

FESWMS Graphical Interface

The FESWMS Graphical Interface includes tools to assist with creating, editing and debugging a FESWMS model. The FESWMS interface exists in the Mesh Module.

Model Control

The FESWMS Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in FESWMS include flows in/out of the model domain or known water surface elevations. In FESWMS boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow). See FESWMS BC Nodestrings and FESWMS BC Nodes for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

Hydraulic Structures

FESWMS was designed for use around highways and includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See FESWMS Hydraulic Structures for more information.

Running the Model

The FESWMS Files are written automatically with the SMS project file or can be saved separately using the *File* | *Save FESWMS* or *File* | *Save As* menu commands. See FESWMS Files for more information on the files used for the FESWMS run.

FESWMS can be launched from SMS using the *FESWMS* | *Run FSTD2H* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *FESWMS* | *Model Check* menu command.

FESWMS Menu

See FESWMS Menu for more information.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- FESWMS Spindown
- Total Flow Nodestring

FESWMS Hydraulic Structures

Overview

FESWMS has the capability to model hydraulic structures including: bridges, roadway embankments, culverts, weirs, and drop-inlet spillways. In the finite element network, bridges and roadway embankments are represented with a collection of two-dimensional elements, which overlay the plan view of these structures. However, since culverts, weirs, and drop-inlet spillways are difficult to characterize with elements, these structures are modeled with either one or two node points, with these nodes representing points of inflow and outflow. An exception to this modeling technique occurs when a culvert spans a large channel or is large in comparison to the size of the defined floodplain elements; in this instance, the user should consider modeling the culvert with two-dimensional elements.

Culvert

A small culvert under a pedestrian walkway.

Small culverts are modeled with inlet and outlet node points in FESWMS unless they are located on the model boundary, in which case they are represented with one node point. Parameters for characterizing small culverts include barrel geometry (shape, length, span, slope, and number of barrels), inlet configuration, barrel roughness, and tailwater depth. FESWMS employs the FHWA [2] culvert routing routines as presented in *Hydraulic Design Series 5 (HDS 5), Hydraulic Design of Highway Culverts* [1] [2] and discharge through a culvert is determined by the energy heads at the nodes where the culvert is placed. To define a culvert with two mesh nodes, proceed with the following steps:

1. Select the culvert's inlet and outlet node by choosing the Select Mesh Node tool and clicking on the nodes of interest while pressing the *Shift* key.
2. Select the menu option *FESWMS | Culvert*. The FESWMS Culvert dialog will appear allowing the entry of culvert parameters. Note that when the Flap-gate option is selected, the model only permits flow to travel in the downstream direction. At this point, the user should verify that the nodes identified as "upstream" and "downstream" in the FESWMS Culvert dialog represent the culvert inlet and outlet, respectively.
3. Culvert material, shape, and inlet type may be designated by selecting the appropriate drop-down menus and clicking on the desired values. Additionally, the FESWMS Culvert dialog offers help in determining the culvert entrance loss coefficient (ke) and the Mannings roughess coefficient of the culvert barrel (nc); simply select the *Help* button next to these two parameters for information. To enter inlet control flow coefficients other than the default values provided (not recommended for novice users), click on the Override defaults box net to the appropriate parameters and enter a new value. Further information concerning the selection of culvert parameters is provided in HDS 5.

Weir

The weir at Coburg Lake in Victoria (Australia) after heavy rainfall.

FESWMS weirs can be used to model flows over topping bridge decks, embankments, guide banks (spur dikes), and other structures. As with culverts, weirs located in the interior of the finite element network are described by two node points, one on the upstream side and one on the downstream side of the weir. Weir flow is typically modeled by dividing the bridge deck, embankment, or other structure into a series of weir segments. Each segment is described by the appropriate number of nodes (either one or two), a discharge coefficient, submergence criteria, and the length and crest elevation of the segment (all necessary as input to the broad-crested weir-flow equation). To define a weir segment, perform the following steps:

1. Select one or two nodes (depending on if the weir is on the boundary or in the interior of the model) by clicking on the Select Mesh Node tool and choosing the node or nodes of interest in defining the weir. If defining a "two-node" weir, hold the *Shift* key to select both the upstream and downstream node.
2. Select the menu option *FESWMS* | *Weir* to open the FESWMS Weir dialog.
3. Adjust the default parameters and enter the user-defined values as necessary. The "paved roadway" option is typically selected from the Weir type drop-down menu for bridge decks. As with culverts, when the Flap-gate option is selected, the model only permits flow to travel in the downstream direction. Also, the Switch button can be used to change the flow direction if the nodes identified as "upstream" and "downstream" do not correspond to the actual flow direction over the modeled weir.

When defining a series of weir segments to model a bridge deck or other structure, one can create nodestrings on the upstream and downstream sides of the structure. After creating the nodestrings, select each nodestring using the Select Nodestring tool in turn while holding the *Shift* key, click on the menu option *FESWMS* | *Weir*, and assign the appropriate crest elevation and other parameters. Further information on weir flow and the broad-crested weir equation can be found in the FESWMS Users Manual [3] and FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [4] [5].

Drop-Inlet Spillways

Drop-inlet spillways are sometimes used as flow conveyance structures in reservoirs, and the discharge through this type of spillway is limited by one of three flow conditions (depending on the existing water surface at the inlet and the configuration of the structure):

1. Weir flow
2. Orifice flow
3. Full pipe flow

Flow through a drop-inlet spillway in the interior of a finite element network is modeled by designating an inflow (entrance) node, an out-flow (exit) node, and a set of parameters that describe the structure. If the structure is located on one of the model boundaries, only an entrace node is specified (as flow is assumed to exit the model domain).

To specify a drop-inlet spillway in the FESWMS model:

1. Select one or two nodes (depending on if the drop-inlet spillway is on the boundary or in the interior of the model) by clicking on the Select Mesh Node tool and selecting the nodes of interst.
2. Select the menu option *FESWMS* | *Drop Inlet*. The Drop Inlet Definition dialog will open, allowing the entry of the inlet characteristics including geometry and discharge coefficients.

Help with selection of weir and orifice parameters is provided in the FHWA publication Hydraulic Engineering Circular (HEC) 22, Urban Drainage Design Manual [4] [5].

Bridges & Roadway Embankments

Unlike culverts, weirs, and drop-inlet spillways, bridges and roadways will usually be modeled with a collection of two-dimensional elements. Bridge modeling considerations include:

1. Deck roughness
2. Weir flow over decking
3. Pressure flow through the bridge opening
4. Flow around bridge piers
5. Embankment elevations

Once bridge elements are created (in the Map Module during network construction) such that they conform to the two-dimensional plan view of the bridge deck, further treatment of the bridge and associated embankments continues as described below:

Deck Roughness

To define the roughness of the bridge deck elements in the Mesh Module, proceed as follows:

1. Click on the Select Element tool and highlight the elements representing the bridge deck while holding the *Shift* key.
2. Select the menu option *FESWMS | Material Properties* and click on the *Roughness Parameters* tab of the FESWMS Material Properties dialog.
3. To define a material property group for the bridge elements (if not previously defined during network construction), select the *General material properties* button, click on the *New* button in the *Materials Data* window, and enter the property name for the bridge elements. Once finished, click *OK*.
4. in the *FESWMS | Material Properties* dialog, highlight the name of the bridge elements in the left-hand window and enter the Manning's n in the *Deck Roughness* edit field. Click *OK*.

A similar procedure can be used to define roughness values for the roadway, including embankments.

Weir Flow Over Decking

Weir flow over decking can be modeled by segmenting the bridge and approaches and establishing a weir for each segment as described previously.

Pressure Flow Through The Bridge Opening

In order for pressure flow to be calculated, the property group representing the bridge deck needs to be coded for potential pressure flow and a bridge deck ceiling elevation (also known as the low chord or low steel elevation) needs to be specified for each element node in the deck as follows:

1. To indicate potential pressure flow, first select *FESWMS | Material Properties* and under the *Roughness Parameters* tab highlight the bridge properties group and select the *Potential Pressure* flow box.
2. Next, code in the ceiling elevation for each element node in the deck by using the Select Mesh Node tool and highlighting the element nodes corresponding to the bridge deck by holding the *Shift* key and clicking on each node to be assigned the ceiling elevation. An alternative means of selecting all bridge nodes is to choose the Select Mesh Node tool , click on the menu option *Edit | Select by Material Type*, choose the *Bridge* material type, and click on *Select*. Once the appropriate bridge deck nodes have been highlighted, select *FESWMS | Local Parameters* and click on the *Ceiling Options* button. In the *Specify ceiling for selected items* box, enter the ceiling or low chord elevation of the selected nodes and click *OK*.

Bridge and Embankment Elevation Adjustments

To adjust bridge and embankment elevations:

1. Focus in on the area surrounding the embankment or bridge nodes of interest using the Zoom tool.
2. Select the corner nodes of the roadway element using the Select Mesh Node tool .
3. Adjust the elevations of the nodes to the desired values by overwriting the z-coordinate value in the Z node coordinate display edit field in the SMS window. Note that adjusting the corner nodes automatically adjusts the center and midside nodes.
4. Repeat these steps until all bridge and roadway elements have been set to the desired elevations.

Piers

Piers may be modeled in SMS by either of two methods:

1. Creating elements of the proper geometric sizes and elevations in the network to mimic piers
2. Utilizing the tools provided in SMS including the FESWMS Pier Definition dialog

To create a pier in the Mesh Module using the FESWMS Pier Definition Dialog:

1. Select the Create Pier tool and click on the approximate position of the pier in the finite element mesh.
2. Choose the Select Pier tool, select the pier of interest, and choose the menu option *FESWMS | Pier*. The *FESWMS Pier Definition* dialog will open. Pier parameters used by FESWMS describe the pier location, geometry, and flow resistance. The pier angle is the orientation of the long axis of the pier measure counterclockwise from the x-axis.
3. Adjust the pier parameters, including coordinate location, as necessary by filling in the various edit fields under *Pier Attributes*. Note that pier names can be changed by clicking on the name displayed in the *FESWMS Pier Definition* dialog.
4. Various parameters for pier scour computations can also be specified by selecting the *Global Options* button. For more detailed information on pier parameters and scour calcuations, refer to FHWA publication Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [6] [7] [8] [9].

Modeling Tips

- When dividing a weir into segments on an element basis, it is recommended that you assign 2/3 of the length to the midside node of an element, and 1/6 of the length to each of the adjacent vertex nodes.
- More than two rows of elements may be required to accurately model pressure flow through a river crossing.
- A smaller, more highly refined finite element mesh should be used at and around piers, abutments, and other areas of curvilinear/complex flow patterns to improve the accuracy of model results.

External Links

- Hydraulic Design Series 5 (HDS 5), Hydraulic Design of Highway Culverts [1] [2]
- Hydraulic Engineering Circular (HEC) 18, Evaluating Scour at Bridges [6] [7] [8] [9].
- User's Manual for FESWMS Flo2DH [3]
- Urban Drainage Design Manual Second Edition [4] [5]

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Model Control Dialog

- FESWMS Spindown

References

[1] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=7&id=13
[2] http://isddc.dot.gov/OLPFiles/FHWA/012545.pdf
[3] http://www.cflhd.gov/design/hyd/flo2dh_manual3.pdf
[4] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=22&id=47
[5] http://isddc.dot.gov/OLPFiles/FHWA/010593.pdf
[6] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=17&id=37
[7] http://isddc.dot.gov/OLPFiles/FHWA/010590.pdf
[8] http://www.fhwa.dot.gov/engineering/hydraulics/library_arc.cfm?pub_number=17&id=38
[9] http://www.fhwa.dot.gov/engineering/hydraulics/pubs/hec/hyderra.cfm

FESWMS Material Properties

Material properties include roughness, turbulence, and wind/wave parameters.

Roughness Parameters

The roughness helps determine the energy losses as water flows over elements. Each material includes roughness information.

- Manning n values (n1, n2, depth1, depth2)

The primary roughness property is the manning n value associated with the element. The manning n value can vary with depth by specifying n values at two depths. The n1 value is used below depth1. The n2 value is used above depth2. Between depth1 and depth2 the n value is linearly interpolated.

- Wall roughness

The wall roughness is used on the edge of the model domain. Wall roughness is ignored unless the model is using semi-slip boundaries.

- Soil Liners

FESWMS can be used with materials representing a soil liner. To use a liner, turn on the linear critical shear stress and set the value. When using a liner n1, n2, depth1, and depth2 are ignored.

- Pressure flow

It is required to toggle on "potential pressure flow" with all materials that are assigned to elements with a ceiling elevation. If pressure flow is enabled, the deck roughness is the manning value for the bridge deck. Otherwise deck roughness is ignored.

- Chezy

Chezy values are an alternative to using Manning n values for roughness. You must turn on chezy in the model parameters to use this value.

- Bed critical shear stress

The bed critical shear stress is used to compute clear water scour.

Turbulence Parameters

Turbulence parameters are used to control the energy lost in turbulence.

- Vo – Base eddy viscosity in Length^2/second
- Cu1, Cu2 – Turbulence model coefficients used to modify base eddy viscosity
- Eddy diffusivity – used to modify base eddy viscosity and is related to the amount of curves in the channel
- Storativity depth – The global storativity depth (specified in the model parameters) can be overriden on a material level by entering a non-zero value.

FESWMS Menu

The following menu commands are available in the FESWMS Menu:

Boundary Conditions

- Assign BC (Boundary Condition)
- Local Parameters
- Initial Conditions

Hydraulic Structures

- Weir
- Culvert
- Drop Inlet
- Channel Link
- Gate
- Pier

Model Parameters

- FLUX String
- Material Properties
- Model Check
- Model Control
- Run FST2DH

FESWMS Model Control Dialog

The FESWMS model control dialog is used to set general simulation options. This document highlights the more commonly used options. Refer to the FESWMS manual[3] for a more detailed description of how these parameters affect the model results.

General Tab

The controls in the general tab include model description, FESWMS version, run controls, input options, and output options.

- Network stamp and BC descriptor

These are comments that are added at the top of the FESWMS input files. The network file (.net) includes only the network stamp but the bc file includes both the network stamp and the bc descriptor.

- Slip conditions

This controls how boundaries are handled by FESWMS. Slip boundaries allows water to move freely along the boundary. No slip fixes the velocity at fixed boundaries to 0.0. Semi-slip allows water to move along boundaries but reduces the velocities.

- Run Type

A simulation can compute hydrodynamics, sediment, or both hydrodynamics and sediment (semi-coupled). It is not recommended to use the sediment only option. Use the semi-coupled option instead.

- Solution Type

This controls whether the simulation will be treated as steady state or dynamic (time dependent). A steady state run requires boundary conditions that do not change and computes a solution that does not change through time.

- Input Options

This controls the files that will be used in FESWMS. An INI (initial conditions) file is used to provide an initial solution for FESWMS. The initial conditions file format is identical to the FESWMS solution file format and a FESWMS solution may be used for an initial conditions file. Initial conditions files can also be created from inside of SMS. INI files are used as hot starts in the spin down process.

- Output Options

The scalar data set file and vector data set files are additional output files written in the EMRL dataset format. Regardless of whether this file is used or not, the datasets water surface elevation, depth, and velocities are written to the .flo solution file. The dataset files will have the same base name as the other project files and have the extensions scl (scalar) and vec (vector). The data written to the data set file can be specified by clicking on the options button. Available scalar data sets include: Depth-averaged velocity magnitude, Unit flow rate magnitude, Froude number, Mechanical energy head elevation, Bed shear stress magnitude, Vorticity magnitude, Wind Speed, Depth of general scour, Scoured bed elevation, Sediment concentration, Sediment volumetric transport rate, Sediment volumetric transport capacity, and Sediment transport rate deficit. Available vector data sets include: Depth-averaged flow velocities, Unit flow rate, Bed shear stress, Tropical cyclone wind velocities, Sediment volumetric transport rates, and Sediment volumetric transport capacities.

Timing Tab

- Relaxation factor

Affects how fast and reliable FESWMS will find a solution. Reducing this value may help FESWMS achieve a solution but may increase runtime.

- Iterations

This is the maximum number of iterations to perform. There will be fewer iterations if the model reaches the convergence parameters before reaching this number of iterations (see parameters tab for convergence parameters).

- Starting time

The time to start a dynamic simulation in hours.

- Run time

The length of the simulation in hours.

- Time step size

The length of time for each time step in hours. FESWMS is an implicit model so the time step size is not dependent upon the courant number.

Parameters Tab

General Parameters

- Water surface elevation

This is the initial water surface elevation for a cold start simulation. This number should always be larger than the elevation of the highest node in the mesh.

- Unit flow convergence

The maximum change of unit flow between iterations that will be considered a acceptable for a converged solution.

- Water depth convergence

The maximum change of water depth between iterations that will be considered acceptable for a converged solution.

- Storativity depth

Storativity allows an element to remain active as long as the water surface elevation is above the highest node value minus the storativity depth. Specifying a non-zero storativity value generally increases the stability of the model.

- Element wetting/drying

Unless this is checked, FESWMS will not allow elements to wet and dry during a simulation.

- Depth tolerance for drying

Indicates the required depth to rewet a dry node. If this value is zero, any depth above the node will rewet a dry element. Using a value greater than zero helps prevent wet/dry excessive wet/dry oscillations that can lead to instabilities.

Print Tab

This tab controls the information that is written to the printed output file (prt).

Sediment Transport

This tab defines the parameters used for sediment transport including bed configuration.

Wind/Storm Conditions Tab

- Condition

Wind conditions include none, a directional wind, or a storm (cyclone or hurricane).

- Wind Parameters

These parameters are used to define the parameters for a directional wind (the same direction magnitude for the entire model domain).

- Storm Parameters

These parameters are used to define a hurricane or cyclone moving across the model domain. The storm parameters such as direction and central pressure are constant throughout the simulation. The effect of a storm surge combined with the wind fields can be defined using the surge parameters toggle. See the FESWMS manual for a description of the input parameters.

- Wave Parameters

FESWMS has the ability to consider wave induced stresses. The wave height ratio is the breaking water wave height to depth ratio. Waves will break when the significant wave height equals the water depth times this ratio.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Spindown

FESWMS Point Attributes Dialog

The FESWMS Feature Point/Node Attributes dialog is used to set the attributes for a feature point / refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each feature point / refine point include:

Attribute Type Frame

- None
- Boundary Conditions – Options button opens the FESWMS Nodal Boundary Conditions dialog
  - FESWMS BC Nodes – FESWMS allows specifying boundary conditions on a nodal basis although it is generally preferable to use boundary conditions on nodestrings. Please refer to the FESWMS manual for information on the nodal boundary condition options.

Options Frame

- Refine point (checked = on)
  - Element size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.

Related Topics

- Feature Objects Menu
- Mesh Generation

FESWMS Polygon Attributes Dialog

The 2D Mesh Polygon Properties dialog is used to set meshing options for the conceptual model. See Mesh Generation for a discussion of the conceptual modeling approach.

The following polygon attributes can be set:

- **Mesh Type** – Specify the mesh generation algorithm. The following options are available:
  - Patch
  - Paving
  - Scalar Paving Density.
- **Bathymetry Type** – Specify the bathymetry source for assignment to the mesh. The following options are available:
  - Constant – assignes a constant elevation value to all nodes in the mesh
  - Scatter Set – interpolates elevation values from the specified scatter set
  - Existing Mesh – interpolates elevation values from an existing 2D mesh
- **Material** – material type to assign to elements created within the polygon
- **Arc Options** – Used to modify the feature vertices along the arc. The arc must be selected in the polygon preview window. The following options are available:
  - Use Original Vertices
  - Distribute Vertices – Change the number of vertices along the arc. A bias can be specified rather than distributing the vertices with a constant spacing.
- **Node Options** – When the mesh type is set to Patch, the node options are used to define the 3 or 4 sides for the patch mesh generation. If a feature point is selected in the polygon preview window, the following options are available:
  - Split – The mesh generation algorithm will treat the two arcs meeting at the feature point as separate sides.
  - Merge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a single side.
  - Degenerate Edge – The mesh generation algorithm will treat the two arcs meeting at the feature point as a "degenerate edge." When using a degenerate edge, the Patching algorithm will require only 3 sides to be defined. This option is only valid for meshes which allow triangular elements. Only one degenerate edge can be specified per feature polygon.

Related Topics

- Feature Objects
- Map Module Menus

FESWMS Sediment Control

Overview

The FESWMS engine was modified in 2005 to include a capacity for sediment transport. In fact at this time the official name of the system changed from Flo2DH to FST2DH. The "ST" stands for "Sediment Transport". That functionality is supported in the SMS interface for the model. Shortly after the capability was added to the numerical engine, a study was performed at Brigham Young University to verify the capabilities and provide sample applications. It was discovered that the FESWMS engine does indeed provide reasonable estimates of sediment transport, including scour and deposition patterns for some applications of non-cohesive sediments. However, the study also revealed that several of the advertised capabilities do not function, and the model is prone to numerical instablities. We advise caution and patience for modelers attempting to use this capability of the FESWMS engine.

Before simulating sediment transport, set up a non-transport project. Then add the sediment tranpsport parameters and attributes. This is done in three main steps:

1. Set the model parameters
2. Define the bed conditions and sediment grain sizes
3. Define the sediment boundary conditions

Model Parameters

A sediment transport simulation requires several settings in the **General** tab of the **FESWMS Model Control** be set. These include the "Run Type", the "FST2DH Output" and the "Solution Type". Set the "Run Type" to "Sediment" or "Semi-coupled". The "Sediment" option uses an existing hydrodynamic solution. The "Semi-coupled" option instructs the engine to switch back and forth between hydrodynamic calculations and sediment calculations at each time step. Tests have shown that for stability, the "Semi-coupled" option should be used. The "Solution Type" should be set to "Dynamic". The "Sediment Transport Solutions" toggle should also be turned on.

The **Sediment Transport** tab of the **FESWMS Model Control** provides access to the other parameters related to sediment transport calculations in FESWMS. The "Report Options" section allows specification of what information will be saved to the report (*.prt) file by FESWMS. The "Control Options" section of the sediment control allows for specification of the numeric properties such as convergence criteria an maximum iterations the engine should use while solving the sediment transport quantities. This section also includes two buttons to acces the sediment parameters and the bed definition options. The parameters include a choice of eight transport formulae. Each formula requires associated variables to be specified.

Bed Conditions

FST2DH performs sediment calculations using a three layer system. The first layer is the "Active bed-layer". This is the layer where deposition and scour can occur. The scour at a single time step is limited to the thickness of the active layer. The second layer consists of material deposited on top of the native bed. If core samples have been taken and show a variation in particle distributions, a depositional-layer may be defined. The third layer defines the native bed. Total scour is limited to the depth or thickness of this layer.

The "Bed Control" button in the sediment control options allows the user to access the interface to define the global bed. This include the thickness of the three computation layers and the distribution of particle sizes in each of those layers. FST2DH supports up to 8 particle sizes. The user defines the characteristic particle size for each gradation and the percent that gradation makes of each of the three layers. If a single size gradation is specified, the three percentages would all be 100%.

The global bed definitions apply to the entire domain unless a local specification is made. This is done by selecting the nodes in the area the local definition changes and define the "Local Paramters" in the FESWMS menu.

Limitation

Tests have shown that numerical sensitivity increases as the number of particle sizes increases. Use as few as you can to represent the situation. Sensitivity also increases with long duration runs (over 1 week).

Sediment Boundary Conditions

Sediment boundary conditions must be applied at all open boundaries just as hydraulic conditions must be defined at those locations. In the case of sediment, the user specifies the amount of sediment entering the domain over that open boundary using one of the following options:

- Don't specify inflow sediment. This means clear water is entering the domain.
- Specify volume of material per time as an essential or natural boundary condition for each particle size.
- Specify concentration (ppm) as an essential or natural condition for each particle size.
- Allow FST2DH to compute the inflow sediments assuming equilibrium conditions.

Limitaion

Tests have shown that only the clear water and equilibrium condition boundary conditions produce reasonable results.

FESWMS Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often FESWMS will not directly converge using these initial conditions.

Incremental Loading Strategy

The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution can be obtained using a series of “Runs” that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called “Hot Starting”. In the incremental loading strategy, “loads” consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

FESWMS Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
  - Water surface elevation
  - Flow rate
- Model Parameters
  - Eddy viscosity

- Finite Element Network
  - Geometry (nodal elevations)

This replaces the need to perform incremental loading by hand in FESWMS.

Related Topics

- FESWMS
- FESWMS Files
- FESWMS Graphical Interface
- FESWMS Hydraulic Structures
- FESWMS Model Control Dialog
- Steering

Saving FESWMS

When you do a *File* | *Save As...* the following files get saved in the *.sms.

- *.fpr referenced to new save location
- *.mat referenced to new save location
- *.flo referenced to original location unless reran then moves to saved location.

6.3. TABS - RMA2/RMA4

TABS

The TABS-MD (Multi-Dimensional) Numerical Modeling System was one of the first widely used collection of programs designed for studying multi-dimensional hydrodynamics in rivers, reservoirs, bays and estuaries. The hydrodynamic engine for the system is the RMA2 engine. RMA2 and RMA4 were written by Resource Management Associates, Lafayette, California, and modified by WES. SED2D was written jointly by Resource Management Associates and WES.

Models

GFGEN

The pre-processor for the TABS software programs. This utility converts ASCII geometry into binary format and does data checking along the way. SMS will launch this utility as needed before running other components of the system.

RMA2

A one-dimensional/two-dimensional numerical model for depth-averaged flow and water levels. RMA2 is a two dimensional depth averaged finite element hydrodynamic numerical model. It computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields. RMA2 computes a finite element solution of the Reynolds form of the Navier-Stokes equations for turbulent flows. Friction is calculated with the Manning’s or Chezy equation, and eddy viscosity coefficients are used to define turbulence characteristics. Both steady and unsteady state (dynamic) problems can be analyzed. It should be noted that the commercially available version of the model does not include all functionality included in the ERDC documentation.

RMA4

A one-dimensional/two-dimensional numerical model for depth-averaged transport. This program uses a provided hydrodynamic solution (either node by node or in RMA2 format) to compute transport of a constituent in solution. It is assumed that the depth concentration distribution is uniform. While model documentation claims up to six constituents can be considered simultaneously, practical application has shown the only application of multiple constituents to be DO/BOD. Either conservative or non-conservative diffusion is computed.

SED2D

Formerly STUDH, a two-dimensional numerical model for depth-averaged transport of cohesive or a representative grain size of noncohesive sediments and their deposition, erosion, and formation of bed deposits. The interface for this model has been removed from version SMS 10.0. Boundary condition files can still be constructed/edited in text editors and SMS can still read solution data.

RMA10

A one-dimensional/two-dimensional/three-dimensional hydrodynamic numerical model. Not available for public use at this time.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website [3].

Related Links

- SMS Models page

External Links

- CHL TABS Numerical Modeling website [1]
- CHL RMA2 Frequently Asked Questions [2]

References

[1] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Software;10
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=ARTICLES;365

TABS Attribute Dialog

TABS Point Attributes Dialog

The *TABS Feature Point/Node Attributes* dialog is used to set the attributes for a feature point / refine point represented by a feature point in a 2D Mesh model coverage. Attributes that can be specified for each feature point / refine point include:

- Refine point (checked = on)
  - Element Size – Specify the nodal spacing, or element edge length in the vicinity of the refine point. Refine points are only used if the mesh is generated using the Paving or Scalar Paving Density mesh generation methods.
- Assign 1-D Geometry – Opens the *GWN Card Definition* dialog

TABS Arc Attributes Dialog

The *TABS Feature Arc Attributes* dialog is used to set the attributes for feature arcs. Attributes that can be specified for each feature arc include:

- Arc Type
  - None
  - Boundary Conditions – Options button opens the RMA2 Nodestring Boundary Conditions dialog
  - Continuity Check/Flux

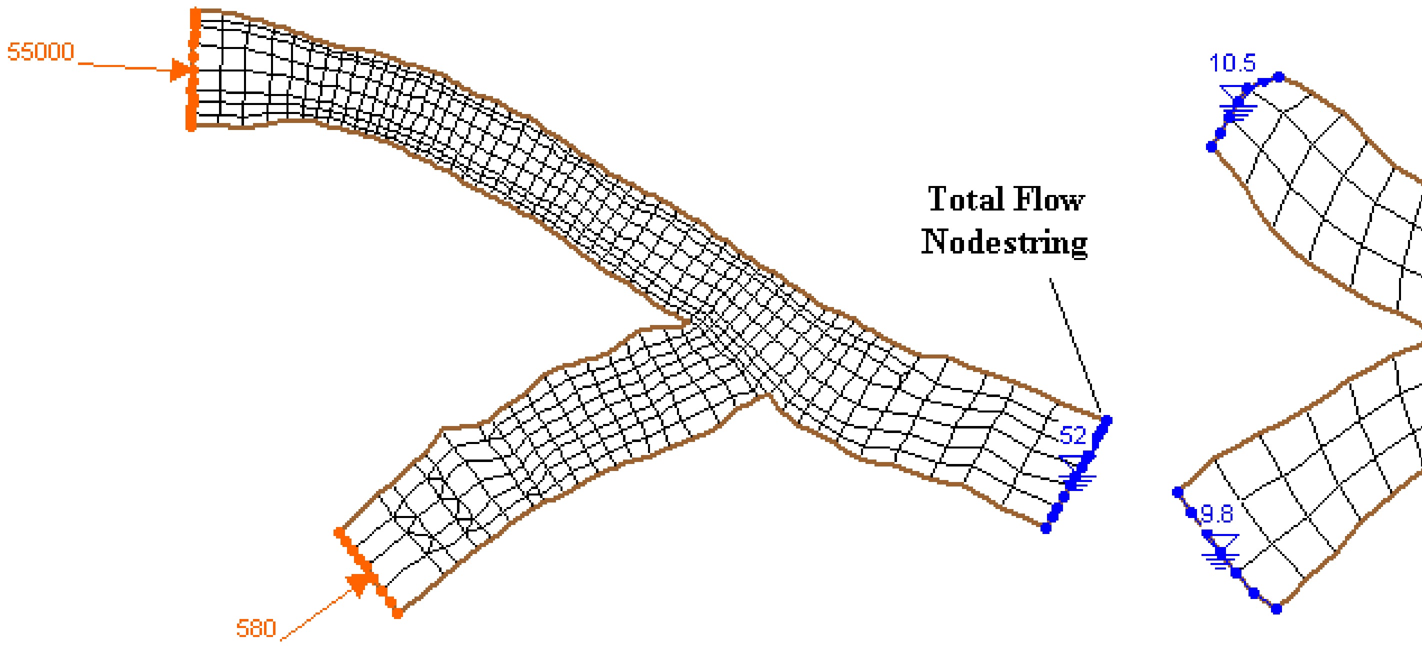
Related Topics

- Feature Objects Menu
- Mesh Generation

Total Flow Nodestring

Two hydrodynamic models supported by SMS, RMA2 and FST2DH, support a concept called the Total Flow Nodestring. These models print out a table of continuity or flux checks across specified nodestrings. Such checks are used to determine if there is mass loss through the model, to determine the amount of flow in various sections of a braided stream, or other similar purpose. In order to participate in the continuity table, the nodestring must either be assigned as a boundary condition or assigned as a flux (FST2DH) or geometric continuity (GC) (RMA2) string.

The nodestring assigned as total flow will always be reported first in the table, and will always have 100% of the flowrate across it. The flux across all other cross sections will be reported in the continuity table as a percentage of this flowrate, whether it be more, less, or equal to 100%. Note that nodestrings should all be created from right to left, while looking downstream. When a nodestring is selected, arrows appear on its ends which show the assumed downstream direction. The figures below show advisable locations of the total flow nodestring.



Related Topics

- Mesh Module

6.3.a. GFGEN

GFGEN

GFGEN (**G**eometry **F**ile **GEN**erator) converts ASCII geometry into binary format and does data checking along the way.

Using the Model / Practical Notes

- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website [3].
- GFGEN Executable Known Issues

External Links

- Users Guide to GFGEN - Version 4.35 [1]

Related Topics

- SMS Models Page
- TABS Models
- RMA2
- RMA4

References

[1] http://chl.erdc.usace.army.mil/Media/3/3/3/gfgen_05_2001.pdf

GFGEN Executable Known Issues

Unfortunately there are some known issues and likely some unknown issues as well related to the GFGEN executable. This page is intended to help make users aware of known issues and how to work around them.

GFGEN (TABS) is developed by Resource Management Associates, Lafayette, California, and modified by WES [1]. Aquaveo does not have access to the GFGEN source code and cannot make changes to fix the issues found in the GFGEN numerical model. Aquaveo can fix problems in the SMS interface, used for pre and post processing of GFGEN files.

This list may not contain all know issues, so please feel free to add to it.

Node Location Truncation

GFGEN does not correctly calculate the midside node location if the x or y coordinate of a node are large. You may receive an error similar to the following:

```
* Node     50 Violates MIDDLE THIRD RULE *****
The node lies       0.0831 of the way from node      1 to node     49
```

```
=== Problem inside COEFS routine. Do 180.
Element= 535 Gauss pt= 1 out of NGP= 16
Divide by DETJ= -0.0229
J11-J22-J12-J21= 1.1526 0.38006 0.74030 0.62257
stop in coefs
```

The current work around is to translate the mesh so the x and y coordinates are smaller values.

See the following forum posts:

- RMA2 - same mesh, same BC, different results?, Results differ depending on nodal coordinates [2]
- Problem inside COEFS routine [3]

Element Limit

GFGEN can only handle up to 100,000 elements.

Related Topics

- GFGEN

References

[1] http://www.wes.army.mil/Welcome.html
[2] http://aquaveo.invisionzone.com/index.php?showtopic=375
[3] http://aquaveo.invisionzone.com/index.php?showtopic=298

6.3.b. RMA2

RMA2

RMA2	
Model Info	
Model type	Two-dimensional depth averaged finite element hydrodynamic numerical model. Computes water surface elevations and horizontal velocity components for subcritical, free-surface flow in two dimensional flow fields.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA2 web site [1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA2 • RMA2 Steering

RMA2 is a 1D/2D hydrodynamic model using the finite element method. RMA2 was written by Ian King and is maintained by the Army Corp of Engineers Engineering Resource Development Center (ERDC) [1]. RMA2 has been applied to multi-dimensional problems since the mid 1970s. As such, it was one of the first widely used multi-dimensional hydrodynamics engine applied to riverine and estuarine applications. The current interface to RMA2 in the Surface-water Modeling System originated as a program named FastTABS.

Functionality

RMA2 has these capabilities:

- Compute water levels and velocities for a 2D mesh structure
- 1D elements (trapezoidal shape)
- 1D flow control structures
- Allow elements to wet and dry
- 2D structures are permitted in version 4.50 or higher (US Army Corps of Engineers employees only)
- Account for effects of the earth's rotation ([[SMS:RMA2_Material_Properties#Coriolis | Coriolis effect]])
- Accepts a wide variety of boundary conditions including:
  - Discharge by node/element/ or line
  - Tidal radiation boundary conditions by line
  - Discharge as a function of elevation by line
  - Water surface elevation along a line
  - Apply wind stress either
    - Uniformly over the model domain; constant or time-varying
    - As a storm; front or tropical cyclonic event (time-varying)

- Read an STWAVE radiation stress file to incorporate wave induced currents
- Identify errors in the computational mesh specification
- Accept either English or standard SI units
- Restart (Hotstart) the simulation from a prior RMA2 run and continue
- Account for Marsh Porosity wetting and drying (wetlands)
- Employ either direct or automatic dynamic assignment of Manning's n-value by water depth
- Employ user selectable manual or automatically assigned turbulent exchange coefficients
- Compute flow across continuity check lines.
- Revisions within a time step (both coefficients and/or boundary conditions)

RMA2 Revisions

RMA2 allows the user to revise both model coefficients and/or boundary conditions for each time step.

Using the Model / Practical Notes

- RMA2 Spindown Steering (Incremental Loading)
- RMA2 2D Control structures are only available to employees of the US Army Corps of Engineers.
- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website [3].
- According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, including storms in simulations remains experimental."
- Boundary condition time series curves are resampled by the model based on the time step. If you change the time step, you may want to re-enter the time series curve since SMS will automatically resample the curve and resampling the curve multiple times may result in a loss of important information.

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the RMA2 model. See RMA2 Graphical Interface for more information.

The RMA2 Graphical Interface contains tools to create and edit an RMA2 simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA2. If a mesh has already been created for a RMA2 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA2. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the RMA2 Menu. See RMA2 Graphical Interface for more information.

Related Topics

- SMS Models Page
- TABS Models
- RMA4
- RMA2 1D Control Structure
- RMA2 Material Properties
- RMA2 Model Control Dialog
- RMA2 Spindown
- RMA2 Graphical Interface
- RMA2 Menu
- RMA2 BC
- RMA2 Files
- Nodal Transition Dialog
- Roughness Options Dialog
- Rainfall Values Dialog

External Links

- CHL RMA2 Website [2]
- CHL RMA2 Frequently Asked Questions [2]
- Users Guide To RMA2 WES Version 4.5 [3]
- Users Guide to RMA2 Version 4.35 [4]
- MERGAVE – Utility for Merging RMA2 Solution Files [5]

References

[1] http://chl.erdc.usace.army.mil/rma2
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=Software;14
[3] http://chl.erdc.usace.army.mil/Media/3/2/7/RMA2_v45_Users_Guide_01-20-2006.pdf
[4] http://chl.erdc.usace.army.mil/Media/3/2/8/rma2_435hlp.zip
[5] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=ARTICLES!427

Nodal Transition (Marsh Porosity) Dialog

The RMA2 Nodal Transition (Marsh Porosity) dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the section "Wetting and Drying", "Elemental Elimination", "Marsh Porosity", and DA and DM Card descriptions in the RMA2 model documentation for more information. The following parameters can be set in the Nodal Transition (Marsh Porosity) dialog:

- Minimum Land Elevation
  - Distance below each node's bathymetry value
  - Constant elevation, independent of nodal bathymetry
- Transition range of distribution
- Minimum wetted surface area factor

Related Topics

- Marsh Porosity
- RMA2 Model Control dialog

Rainfall Values Dialog

The RMA2 Rainfall Values dialog is used to assign rainfall or evaporation. Rainfall or evaporation is applied in inches/hour or cm/hour if using metric units. Positive values represent rainfall, while negative values represent evaporation. Rainfall and evaporation can occur at any time during the simulation. See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation for more information.

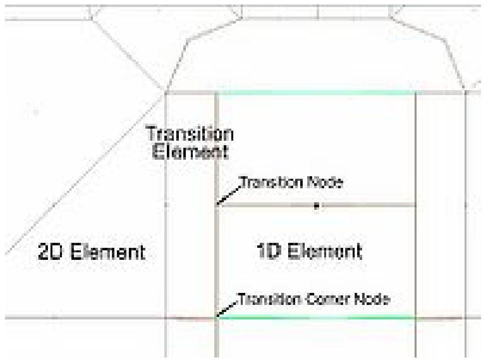
Related Topics

- RMA2 Model Control dialog

RMA2 1D Control Structure

Overview

In RMA2, a one-dimensional Control Structure element is a single point which contains two nodes and has an IMAT value ≥ 904. The order of the node numbering at a Control Structure element should be that the side with higher elevation comes first, then the side with the lower elevation. This is generally the “upstream” side of the structure followed by the “downstream” side. This order sets the “orientation” of the flow through the structure.



Example RMA2 transition element from 2D mesh to 1D element.

Since flow through and over some structures does not fit the 2D assumptions, some structures can be represented as 1D components in the 2D mesh. These include weirs, culverts, drop inlets and some gates.

There are strict rules that must be followed in order to get convergence with 1D elements in RMA2. These rules include:

- Transition elements must be connected to the free end of a 2D element. The element should not be connected to other 2D elements on the sides.
- The transition element must be perpendicular to the free end of the 2D element.
- The bottom elevation of the free end should be flat.
- Control structure elements cannot be constructed on transition elements.

Culvert

A culvert is a conduit used to enclose a flowing body of water. It may be used to allow water to pass underneath a road, railway, or embankment, for example. Culverts can be made of many different materials; steel, polyvinyl chloride (PVC) and concrete are the most common.


A small culvert under a pedestrian walkway.

Culvert flow rate depends primarily on headwater depth. Other factors that affect flow rate are tailwater elevation and properties of the culvert such as shape, cross sectional area, inlet geometry, length, slope, and roughness.

Weir

The weir at Coburg Lake in Victoria (Australia) after heavy rainfall.

A weir is a small overflow-type dam commonly used to raise the level of a river or stream. Weirs have traditionally been used to create mill ponds in such places. Water flows over the top of a weir, although some weirs have sluice gates which release water at a level below the top of the weir. The crest of an overflow spillway on a large dam is often called a weir.

Weirs also give hydrologists and engineers a simple method of measuring the rate of fluid flow in small to medium sized streams. Since the geometry of the top of the weir is known, and all water flows over the weir, the depth of water flowing over the weir will be an indication of the flow. There are different types of weir. It may be a simple metal plate with a V notch cut into it or it may be a concrete and steel structure across the bed of a river. A v-notch weir will give a more accurate indication of low flow rates.

Related Topics

- RMA2

RMA2 BC

Nodestring BC

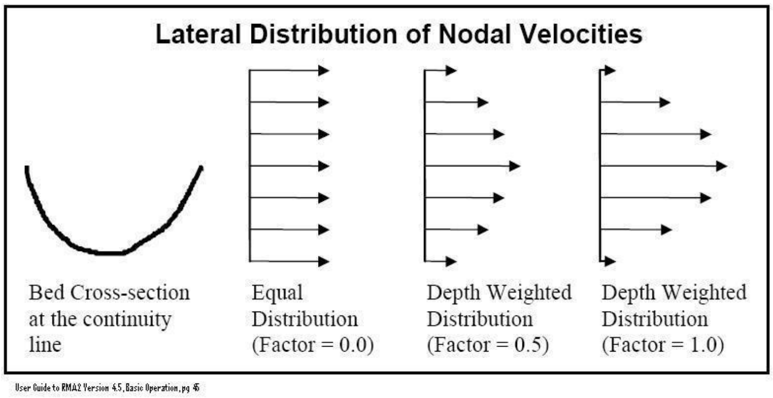
Boundary Condition Type

None

No Boundary Condition is assigned to this Nodestring

Specified Flow Rate

A flow rate BC is assigned to this nodestring. It is the most popular way to provide a total flow across a particular continuity line. The total flow (cubic units per second), flow angle (degrees from positive x-axis) and distribution factor can be specified.



Water Surface Elevation

A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. (However RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations)

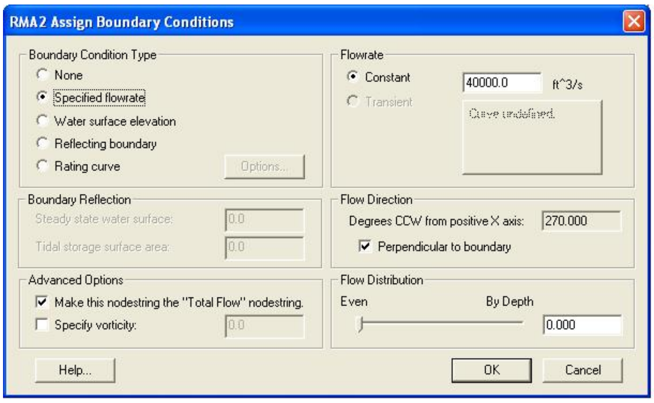
Reflecting Boundary

Considered an advanced feature - for more information see the "Boundary Permeability (Reflection/Absorption)" section on page 99 of the Users Guide to RMA2 version 4.5

Rating Curve

A Rating curve can also be defined by selecting the "Options..." button

RMA2 Nodestring Boundary Conditions Dialog



Node BC

Boundary Condition Type

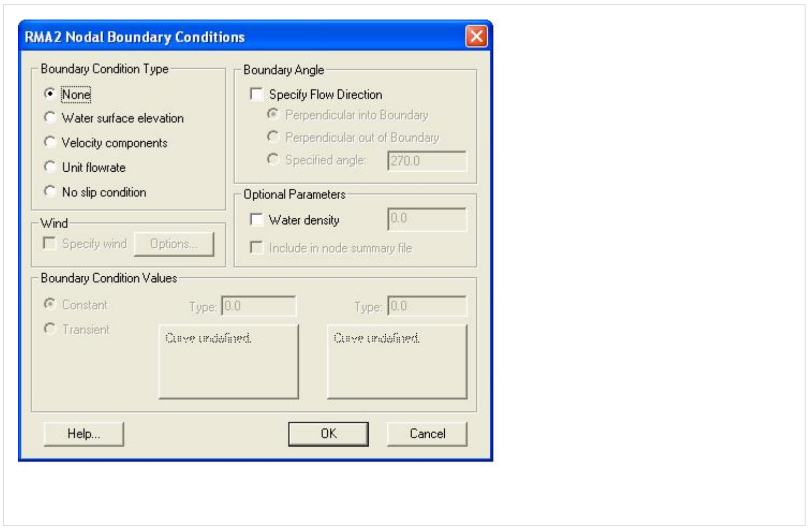
None

No Boundary Condition is assigned to this Nodestring

Water Surface Elevation

A Water surface elevation BC is assigned to this Nodestring. This boundary condition is normally located at the tailwater location. (However RMA2 does sometime edit the water depth at the exit boundary to satisfy the finite element equations)

No Slip Condition



RMA2 Boundary Conditions

Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow).

Nodal Boundary Conditions

See RMA2 Boundary Condition Nodes for more information.

Nodestring Boundary Conditions

See RMA2 Boundary Condition Nodestrings for more information.

Related Topics

RMA2

RMA2 Files

Input and Output files for RMA2.

Input Files

- ASCII Geometry (*.geo) → GFGEN → Binary Geometry (*.bin) and printed output file (*.ot1)
- Run Control Input File (*.bc)
- Hotstart File (*.hot) – optional
- Alternate BC File (*.abc)
- Run File (*.run) – _gr.run (GFGEN) and _rm.run (RMA2)

Output Files

- Binary Solution (*.sol)
- Printed Output File (*.ot2)
- Node summary file (*.sum)
- Hotstart Output (*.hot) – optional

Related Topics

- RMA2
- RMA2 Hydraulic Structures
- RMA2 Model Control Dialog
- RMA2 Spindown

RMA2 Graphical Interface

The RMA2 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA2 model. The RMA2 interface exists in the Mesh Module. Users can visualize and create projects, easily modify project parameters, and view the solutions produced by the RMA2 model. Below are links describing the SMS interface that interacts with the RMA2 model.

Model Control

The RMA2 Model Control Dialog is used to setup the options that apply to the simulation as a whole. These options include time controls (steady state/dynamic), run types, output options, global parameters, print options and other global settings.

Boundary Conditions

All numeric models require boundary condition data. Boundary conditions in RMA2 include flows in/out of the model domain or known water surface elevations. In RMA2 boundary conditions are generally defined on nodestrings but may also be defined on nodes. The default boundary condition is a closed boundary (no flow). See RMA2 BC Nodestrings and RMA2 BC Nodes for more information.

Material Properties

Each element is assigned a material type. Material properties describe the hydraulic characteristics of each material type.

RMA2 Hydraulic Structures

RMA2 includes support for several different types of structures including weirs, culverts, drop inlets, gates and piers. See RMA2 Hydraulic Structures for more information.

Running the Model

The RMA2 Files are written automatically with the SMS project file or can be saved separately using the *File* | *Save RMA2* or *File* | *Save As* menu commands. See RMA2 Files for more information on the files used for the RMA2 run.

RMA2 can be launched from SMS using the *RMA2* | *Run FSTD2H* menu command. A check of some of the common problems called the Model Checker is done each time the model is launched, or by selecting the *RMA2* | *Model Check* menu command.

RMA2 Menu

See RMA2 Menu for more information.

Related Topics

- SMS Models Page
- TABS Models
- RMA2
- RMA4
- Total Flow Nodestring

RMA2 Material Properties

For an in depth discussion of the RMA2 material properties, consult the RMA2 Users Guide. [3] [4]

Turbulence

The global turbulence values (specified in the RMA2 model control) can be overridden for each material.

- Standard eddy viscosity method
  - E_{xx} – The x-momentum of turbulent exchange in the x-direction (lb-sec/ft^2 or Pascal-sec for SI-units)
  - E_{xy} – The x-momentum of turbulent exchange in the y-direction (lb-sec/ft^2 or Pascal-sec for SI-units)
  - E_{yx} – The y-momentum of turbulent exchange in the x-direction (lb-sec/ft^2 or Pascal-sec for SI-units)
  - E_{yy} – The y-momentum of turbulent exchange in the y-direction (lb-sec/ft^2 or Pascal-sec for SI-units)
  - Note: 1 lb-sec/ft^2 = 47.879 Pascal-sec
- Smagorinski method
  - TBFACT – Smagorinsky coefficient for turbulent exchange. A negative value applies the default coefficient (0.05).
  - TBFACTS – Not applied in RMA2. Smagorinsky coefficient for diffusion (Vorticity). A negative value applies the default coefficient (0.05).
  - TBMINF – Smagorinsky minimum turbulent exchange factor. A negative value applies the default coefficient (1.0).
  - TBMINFS – Not applied in RMA2. Smagorinsky minimum diffusion (Vorticity) factor. A negative value applies the default coefficient (1.0).

Roughness

The global roughness value (specified in the RMA2 model control) can be overridden for each material. The material roughness can be specified as a single value, or depth dependent.

Marsh porosity

The global marsh porosity values (specified in the RMA2 model control) can be overridden for each material. Since wetting and drying cause abrupt changes to the shape of the finite element mesh, numerical stability problems can result when wetting and drying occurs. In an attempt to address the fact that wetting and drying is a continuous process, while the mesh is a discreet data set, marsh porosity was implemented for RMA2. When marsh porosity is used, elements are considered "dry" only if ALL nodes on the element are dry.

Abrupt changes in the marsh porosity parameters between adjacent node can lead to divergence. The results from a simulation using marsh porosity can be difficult to interpret due to the fact that partly dry marsh elements appear completely wet even thought the water surface elevation is lower than part of the element.

Marsh porosity is only available when running a dynamic RMA2 simulation.

Wind

According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

Wind shear stress at the water surface is caused by friction between the moving air and water. The shear stress coefficient is a function of wind speed. The global wind values (specified in the RMA2 model control) can be overridden for each material.

Wind parameters are only available when running a dynamic RMA2 simulation.

Rainfall

The global rainfall and evaporation values (specified in the RMA2 model control) can be overridden for each material.

Coriolis

The global coriolis [1] values (specified in the RMA2 model control) can be overridden for each material.

Related Topics

RMA2

References

[1] http://en.wikipedia.org/wiki/Coriolis_effect

RMA2 Menu

The following menu commands are available in the RMA2 Menu:

Boundary Conditions

- Assign BC – Specify boundary conditions for the selected node(s) or nodestring(s)
- Delete BC – Deletes the boundary condition assigned to the selected node(s) or nodestring(s)
- Extract Nodal BC – Extracts nodal boundary conditions (dynamic simulations only)

Hydraulic Structures

- 1D Node Geometry – Specify 1D node geometry (1D Control Structures).

Model Parameters

- Add GC Strings – Designates the selected nodestrings as Geometry Continuity Check Line nodestrings
- Material Properties – Specify turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material
- Model Check – Checks the RMA2 simulation for common input errors
- Model Control – Organize inpute files, specify model parameters, choose output options, etc.
- Revisions – Create and organize revisions
- Run RMA2 – Launches the RMA2 model

RMA2 Model Control Dialog

This article highlights frequently used RMA2 model control parameters. Please see the RMA2 model documentation for a description of items not discussed or for more information on specific items.

RMA2 model parameters are divided into the following groups in the RMA2 Model Control:

General

Simulation Titles / $T1 Card

The RMA2 .geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Machine Type / $M Card

Set the machine type the RMA2 simulation will be run on. Options include:

- Microprocessor (PC) - Default option
- Prime mini-computer
- DEC VAX
- Cray or Cyber-205
- HP or Alpha workstations

Water Properties

Set the properties of the water being simulated.

- Temperature / FT Card
- Density / FD Card
- Coldstart initial water surface / IC Card
- 1D node initial conditions
  - Minimum depth / IC Card
  - Initial velocity / IC Card

Scale Factors

- X scale / GS Card
- Y scale / GS Card
- Z scale / GS Card

Timing

Simulation Type

- Steady state – Boundary conditions do not change with time. A "snapshot" in time.
- Dynamic – Boundary conditions change with time. Computation Time parameters will need to be specified.

Iterations For Flow Calculations

See the TI Card description in the RMA2 model documentation.

- Initial solution – Maximum number of iterations to perform for the initial solution.
- Each time step – Used for dynamic simulation - maximum number of iterations to perform after the first time step.

Computation Time

See the TZ Card description in the RMA2 model documentation.

- Time step size – DELT Variable
- Number of time steps – NCYC Variable
- Maximum time – TMAX Variable
- First time step – NSTART Variable
- Perform intermediate restart – MBAND Variable

Bendway Correction

Used to supply values associated with the calculation of vorticity. See the VO Card description in the RMA2 model documentation.

- Compute vorticity – IVOR Variable
- ASEC – ASEC coefficient for the vorticity equation. Recommend: 5.0
- DSEC – DSEC coefficient for the vorticity equation. Recommend: 0.5
- RCMIN – Minimum radius of curvature that will be allowed. Recommend: 6 feet (or 2 meters).

See also BV card, TV card.

Depth Convergence Parameters

- Steady state depth convergence
- Dynamic depth convergence

Vorticity Convergence Parameters

See the VO Card description in the RMA2 model documentation.

- Steady state vorticity convergence
- Dynamic vorticity convergence

Iterations for Vorticity Calculations

The maximum number of iterations allowed for the vorticity calculation is set by tvariables NVITI or NVITN on the TV card. If the number of passes between the phases reaches NPASS1 or NPASS2, then the calculation has failed to converge. If the number of "good passes" between the two phases exceeds NGOODMAX, the calculation is finished and the code can proceed to the next time step. See the "Bendway Correction (Vorticity)" section and TV Card description in the RMA2 model documentation.

The following TV Card variables are used to control the number of iterations for vorticity calculations:

- Steady state passes – NPASS1 Variable
- Steady state vorticity iterations – NVITI Variable
- Dynamic passes – NPASS2 Variable
- Dynamic vorticity iterations – NVITN Variable
- Number of good passes required – NGOODMAX Variable

Files

RMA2 Input Files

- Specify geometry file – select an existing RMA2 ASCII Geometry (*.geo) file to use with the simulation
- Hotstart input file – specify an existing RMA2 Hotstart File (*.hot) to use with the simulation
  - Use specific time for hotstart
- Alternate dynamic BC file – See the "RMA2 Alternate Dynamic BC File Format" section of the RMA2 model documentation
- Write dynamic memory file – See the "Dynamic Memory Allocation" section of the RMA2 model documentation

RMA2 Solution Files

- Write RMA2 solution file
  - Write frequency in timesteps
- Write hotstart file – Write an RMA2 Hotstart Output (*.hot) file. See RMA2 Spindown for a discussion of hotstarting.
  - Save only the last timestep – Write only the last successful RMA2 timestep solution to the hotstart file.
  - Write frequency in timesteps – Write multiple timesteps to the hotstart file.

Informational Files

- Echo card input to screen
- Write ASCII results file
  - Do not echo any node/element input data
  - Echo all input data except initial conditions
  - Echo all input data
  - Write frequency of nodal results in timesteps
  - Trace subroutines level
- Write summary by node file
- Write automatically computed parameters file
- Write vorticity file
  - Write frequency in timesteps

Global Methods

Global Wet/Dry Technique

See the Material Properties article for a description of the Marsh Porosity parameters.

- Elemental wet/dry check
  - Check frequency in timesteps
  - Dry depth
  - Active depth
- Nodal Transition (Marsh Porosity)
  - Options – Opens the Nodal Transition (Marsh Porosity) dialog

Global Roughness Assignment

- Default roughness value
- Roughness by depth
  - Options – Opens the Roughness Options dialog

Coriolis Forces

- Latitude

Global Eddy Viscosity Assignment

See the Material Properties article for a description of the Turbulence parameters.

- Traditional eddy viscosity approach (default)
- Peclet Number
- Smagorinski Method
  - Exx ratio
  - Eyx ratio
  - TBFACTS
  - TBMINFS
  - Peclet Number
  - Minimum velocity

Weather

Wind

According to the RMA2 model documentation, "Because of lack of experience in using storms in a simulation, this feature remains experimental."

See the "Simulating With Storms" section and BWC Card description in the RMA2 model documentation for more information.

- Method for wind
  - Do not use wind
  - Original RMA2 Formula
  - Van Dorn Formula
  - Wu Formula
  - Safaie Formula
  - Ekma Formula
  - Generic Formula
- A
- exp
- C
- rho
- Specify global wind
  - Define

Rainfall/Evaporation / RA Card

See the "Adding Rainfall And Evaporation" section and RA Card description in the RMA2 model documentation for more information.

- Specify rainfall or evaporation
  - Define – Opens the Rainfall Values dialog

Related Topics

- RMA2

RMA2 Spindown

For cold start simulations, the initial velocities are zero and the water surface elevation is constant. This is often referred to as the "bathtub condition." Often RMA2 will not directly converge using these initial conditions.

Incremental Loading Strategy

The flow equations are nonlinear and thus require an iterative solution, starting from some initial guessed value. Convergence of the iterative solution is not guaranteed. Since the desired boundary conditions may be vastly different from a cold start condition, it may be impossible to get convergence starting from this bathtub type condition. However, a solution can be obtained using a series of “Runs” that generate solutions progressively closer to the desired answer. Intermediate boundary conditions that are closer to the final desired boundary conditions are specified to generate a set of flow conditions. These conditions do not represent the final desired flow conditions, but are closer to the final desired flow conditions than the original cold start, and can be used as initial conditions for a subsequent run. Starting the model from a previous solution is called “Hot Starting”. In the incremental loading strategy, “loads” consisting of applied flow rates and water surface elevations along the boundary increment from a cold start condition to the final condition. By choosing a suitably small increment in the boundary conditions, convergence can be attained.

RMA2 Spindown refers to the process of using the Steering Module to automate the process of incremental loading. The Steering Module can vary the following:

- Boundary Conditions
  - Water surface elevation
  - Flow rate
- Model Parameters
  - Eddy viscosity
- Finite Element Network
  - Geometry (nodal elevations)

This replaces the use of "REV" cards in RMA2.

Related Topics

- RMA2
- Steering

Roughness Options Dialog

The RMA2 Roughness Options dialog is used to define the RD Card: Automatic Roughness Coefficient Assignment by Depth. See the RD Card description in the RMA2 model documentation for more information.

The following parameters can be set in the Roughness Options dialog:

- Manning's N, no vegetation
- Depth for no vegetation
- Manning's N, with vegetation
- Roughness coefficient
- Minimum Manning's N
- Maximum Manning's N
- Set Defaults
  - Opens the Roughness By Depth Defaults dialog. Defaults from several Army Corps projects can be automatically entered into the Roughness Options dialog.

A graph of the Depth vs. Roughness is calculated using the given parameters and shown in the Roughness Options dialog.

Related Topics

- RMA2 Model Control dialog

Saving RMA2

When you do a *File* | *Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.sim referenced to new save location

6.3.c. RMA4

RMA4

RMA4	
Model Info	
Model type	Finite element water quality transport numerical model in which the depth concentration distribution is assumed uniform. It computes concentrations for up to 6 constituents, either conservative or non-conservative, within the computational mesh domain.
Developer	Resource Management Associates United States Army Corps of Engineers (USACE)
Web site	RMA4 web site [1]
Tutorials	General Section • Mesh Editing • Observation • Overview • Sensitivity Models Section • RMA4

RMA4 is a finite element water quality transport numerical model. RMA4 was designed to simulate the depth-average advection / diffusion process in an aquatic environment. The model can be used for the evaluation of any conservative substance that is either dissolved in the water or may be assumed to be neutrally buoyant within the water column. The model is also used for investigating the physical processes of migration and mixing of a soluble substance in reservoirs, rivers, bays, estuaries and coastal zones. The model is useful for evaluation of the basic processes or for defining the effectiveness of remedial measures. For most applications, the model utilizes the depth-averaged hydrodynamics from RMA2. The input hydrodynamics solution can be either a steady state or a dynamic simulation and is used to compute particle dispersions.

Functionality

RMA4 has been applied to:

- Define horizontal salinity distribution.
- Trace temperature effects from power plants
- Calculate residence times of harbors or basins
- Optimize the placement of outfalls
- Identify critical area for oil spills or other pollutants spread
- Monitor water quality criterion within game and fish habitats
- Determine the limits of salinity intrusion
- Perform flushing analysis
- Turbidity monitoring
- SMS does not support BOD/DO modeling with RMA4 because of issues with RMA4.

Using the Model / Practical Notes

- RMA4 includes support of the SI card which controls what units the engine will use. If SI is set to 0, the model runs with English units. If it is set to 1, the model runs in metric units. SMS saves the SI card based on the coordinate system (projection) specified by the user. However, it is recommended that metric units be used for RMA4. This applies to the hydrodynamics that are fed into RMA4 as well.
- RMA4 is more sensitive to mass gain/loss than RMA2 across large closed boundary angles. The suggested minimum boundary angle between any two adjacent elements is 10°. This becomes a very great concern, as this applies simply to the wet/dry interface and not to the outer mesh boundary.
- RMA4 should not be used to evaluate highly volatile materials such as gasoline, nor materials that do not mix with water, such as oil. Although RMA4 has been used to measure temperature effects from such locations as power plant discharges, this is not a recommended application. Although the documentation claims to be able to model up to 6 constituents, the SMS developers have never had success with more than a single constituent at a time. Multiple constituents require multiple boundary condition files, each with values for one of the constituents. Multiple constituents can be named and joined through the Data Calculator and exported to a single file for future use.
- Card order matters. The RMA4 run control and full print files are very helpful for determining what options are being used.
- The TABS models are built to expire after a set date. You can download the latest version from the Software Updates section of the Aquaveo website [3].

Graphical Interface

SMS provides a graphical interface that is designed to allow users to visualize the projects they are creating, easily modify project parameters, and view the solutions produced by the RMA4 model. See RMA4 Graphical Interface for more information.

The RMA4 Graphical Interface contains tools to create and edit an RMA4 simulation. The simulation consists of a geometric definition of the model domain (the mesh) and a set of numerical parameters. The parameters define the boundary conditions and options pertinent to the model.

The interface is accessed by selecting the 2D Mesh Module and setting the current model to RMA4. If a mesh has already been created for a RMA4 simulation or an existing simulation read, the mesh object will exist in the Project Explorer and selecting that object will make the 2D Mesh module active and set the model to RMA4. See the Mesh Module documentation for guidance on building and editing meshes as well as visualizing mesh results.

The interface consists of the 2D Mesh Module Menus and tools augmented by the RMA4 Menu. See RMA4 Graphical Interface for more information.

Related Topics

- SMS Models Page
- TABS Models
- RMA2

External Links

- CHL RMA4 Frequently Asked Questions [2]

References

[1] http://chl.erdc.usace.army.mil/rma4
[2] http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=ARTICLES;366&g=74

RMA4 BC Elements

A boundary mass loading can be added to an element in the mesh by selecting first the desired element and then the RMA4 | Assign BC menu command. This opens the RMA4 Assign Element Mass Loading dialog where a constant or transient entry can be made in grams/second for each constituent defined in the RMA4 model control dialog. The time variant option uses a XY Series File, XYS Format editor to enter a series of time step-mass loading pairs. The time steps should be consistent with those of the RMA4 simulation and for each constituent. The mass loadings may be deleted using the RMA4/Delete BC menu command.

Card Type	**RMA4 BLE Card: Boundary Loading (Mass)**		
Description	Used to assign mass loading on the element boundary.		
Required	NO		
Format	BLE		
Sample	BLE 237 2 3		
Field	**Variable**	**Value**	**Description**
0	IC1	BLE	Card type identifier.
1	ISTART	+	Element id
2	PBCX(J,1)	+	Mass loading for constituent 1 at ISTART=J (g/s)
3	PBCX(J,2)	+	Mass loading for constituent 2 at ISTART=J (g/s)
4	PBCX(J,3)	+	Mass loading for constituent 3 at ISTART=J (g/s)
5	PBCX(J,4)	+	Mass loading for constituent 4 at ISTART=J (g/s)
6	PBCX(J,5)	+	Mass loading for constituent 5 at ISTART=J (g/s)
7	PBCX(J,6)	+	Mass loading for constituent 6 at ISTART=J (g/s)

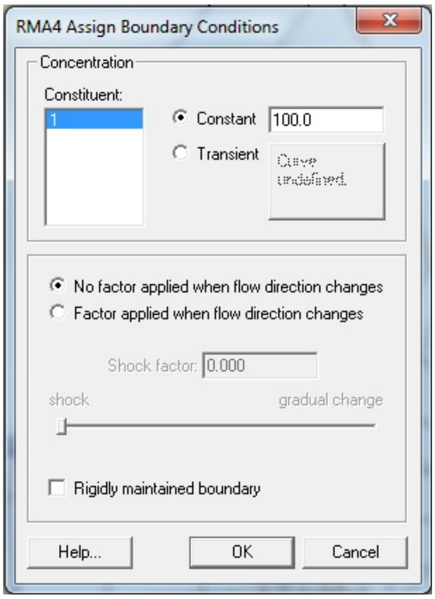
Card Type	END
Description	Identifies the end of the time step
Required	NO

Related Topics

- RMA4
- RMA4 Boundary Conditions

RMA4 BC Nodestrings

Nodestrings can be used as boundary conditions to define constituent concentrations being modeled. Constituent concentrations can be defined as constant or transient using the XY Series Editor.



RMA4 Assign Boundary Conditions dialog

RMA4 does not care about the units of the concentration because the output is relative to the initial number you specify. For example, if we specify a concentration of 1,000, the values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration is ppm, ppt, or kg/kg; RMA4 treats all concentrations as relative values.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

RMA4 Boundary Conditions

Assigning Boundary Conditions

Boundary conditions are generally defined on nodestrings but may also be defined on elements. The default RMA4 boundary condition is a closed boundary (no constituent). To assign boundary conditions, select the item (nodestring, element, etc.) to which the boundary condition should be applied and use the menu command RMA4 | Assign Boundary Conditon which will open a dialog allowing the specification of boundary condition options.

Deleting Boundary Conditions

Boundary conditions are removed using the menu command RMA4 | Delete Boundary Conditon

Nodestring Boundary Conditions

See RMA24 Boundary Condition Nodestrings for more information.

Element Boundary Conditions

See RMA4 Boundary Condition Elements for more information.

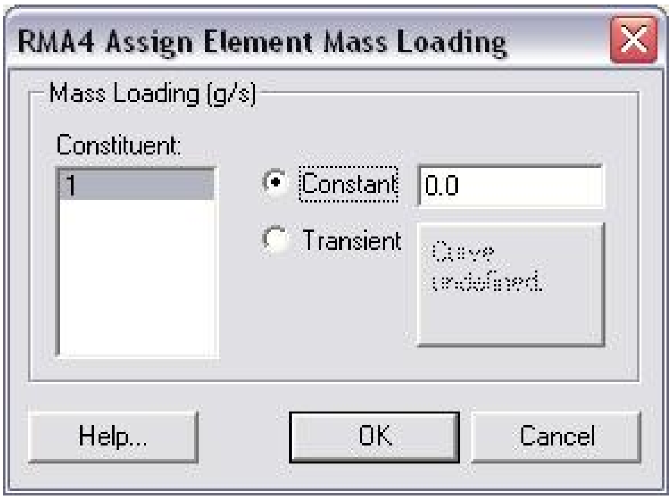
Related Topics

- RMA4
- RMA2 Boundary Conditions

RMA4 Element Loading

Starting in SMS 10.1, it is now possible to assign a mass loading to an element.

To do this, select the element then go to RMA4 | Assign BC.



RMA4 Files

Input and Output files for RMA4.

Input Files

- Hydrodynamic Solution (usually an RMA2 *.sol file with a path name less than 67 characters long)
- RMA4 Run Control (Boundary Condition) input (*.trn)
- GFGEN Output Geometry (*.bin)

Output Files

- Transport Solution (*.qsl)
- Full Print Listing File (*.ot3)

Related Topics

- RMA4

RMA4 Graphical Interface

The RMA4 Graphical Interface includes tools to assist with creating, editing, and debugging a RMA4 model. The RMA4 interface exists in the Mesh Module. Users can visualize and create projects, easily modify project parameters, and view the solutions produced by the RMA4 model.

The following links to describe the SMS interface that interacts with the RMA4 model:

- RMA4 Menu

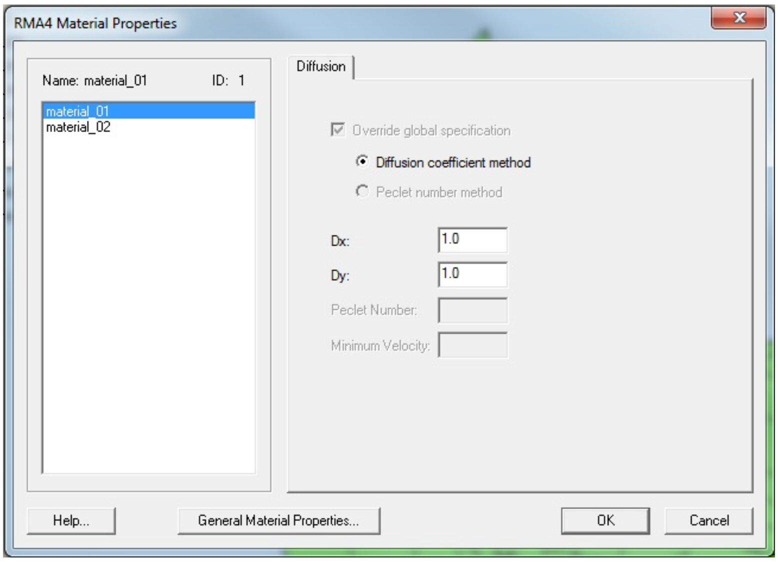
Related Topics

- TABS Models
- GFGEN
- RMA2
- RMA4

RMA4 Material Properties

Diffusion

Because RMA4 does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the x and y directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.



RMA4 Material Properties dialog

RMA4 Menu

The following menu commands are available in the RMA4 Menu:

Boundary Conditions

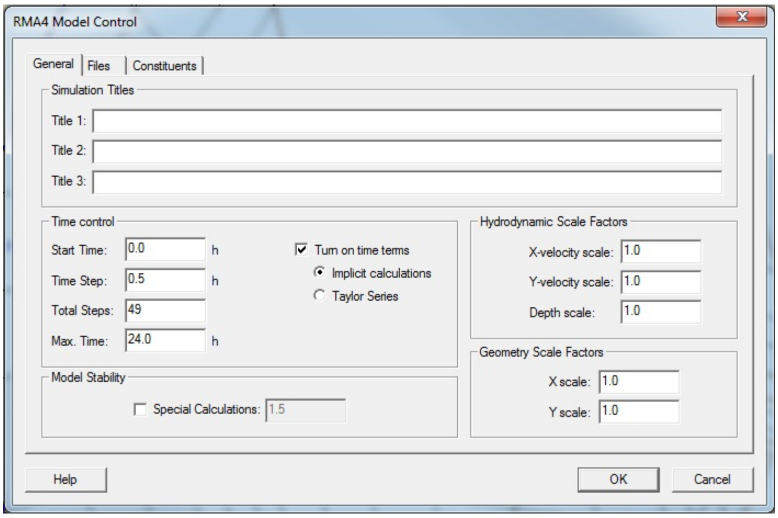
- Assign BC – Specify boundary conditions for the selected nodestring(s) or element(s)
- Delete BC – Deletes the boundary condition assigned to the selected nodestring(s) or element(s)

Model Parameters

- Delete Simulation – Deletes the current RMA4 simulation. Sets all values in the Model Control to default values.
- Material Properties – Specify turbulence, roughness, marsh porosity, wind, rainfall, and coriolis properties for each material
- Model Control – Organize inpute files, specify model parameters, choose output options, etc.
- Run RMA4 – Launches the RMA4 model

RMA4 Model Control

The Model Control Dialog is used to define general, file, and constituents characteristics. The General tab includes options to set up the period of time the model will run, and hydrodynamic and geometry scale factors. The File tab includes the time step used from the RMA2 velocity file and the time subtracted from the RMA2 velocity file, RMA4 Output files options, and Informational Files to be produced when running the model. The Constituents tab includes diffusion coefficient for the constituent, constituent control, and mass check.



RMA4 Model Control

General

Simulation Titles. The RMA4 .geo file contains three comment lines, which can be set using the Title 1, Title 2, and Title 3 edit fields.

Time control

Start time

Time Step

Total Steps

Max. Time

Hydrodynamic Scale Factors

X-Velocity scale

Y-Velocity scale

Depth scale

Geometry Scale Factors

X scale

Y scale

Files

RMA4 Input Files. Time steps from the RMA2 solution file that will be used for the RMA4 simulation.

RMA4 Output files. Output files when running RMA4

Information Files. Files to create when running RMA4

Constituents

Diffusion Coefficient. Diffusion Coefficient Method and Peclet Number Method.

Constituent Control. Number of constituents, initial concentration for each of the constituents, and decay coefficient for each coefficient.

Mass Check. Mass conservation check.

Saving RMA4

Return to saved location page

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- *.mat referenced to new save location
- *.qsl referenced to new save location (RMA4 solution file)
- *.sol referenced to new save location and original location (RMA2 solution file)

6.4. TUFLOW

TUFLOW

TUFLOW	
Model Info	
Model type	One-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.
Developer	Bill Syme BMT WBM (Australia) [1]
Web site	TUFLOW web site [2]
Tutorials	Models Section • TUFLOW 2D • TUFLOW 1D

TUFLOW ENGINE

The TUFLOW computational engine computes 1D and 2D hydraulic solutions. The engine is very stable making it an excellent choice for models with lots of wetting and drying. TUFLOW includes 1D cross-sections, 1D pipe networks, 1D and 2D hydraulic structures, and allows rainfall directly on the model domain so it is useful in many different applications including coastal, riverine and urban settings.

For more information see the TUFLOW webpage www.tuflow.com [3].

SMS Interface

The TUFLOW Interface in SMS can be used to construct TUFLOW models and view and analyze the results.

Some of the features of the interface include:

- Create 2D domains from bathymetry data
- Extract 1D cross-section domains from TIN data
- Define 1D pipe domains including links to the surface
- Import data from ArcGIS or MapInfo GIS formats
- Define boundary conditions at 1D nodes, 2D lines, or 2D polygons
- Create 2D geometry modifications to model objects such as levees
- Set material properties such as Manning n values and hydrologic losses
- Define simulations from building blocks above
- Simulations share common data to prevent data duplication errors and make it easy to update project with new data

The interface consists of the Cartesian grid tools, the TUFLOW menu and the TUFLOW coverages.

Using the Model / Practical Notes

- For the past several years, TUFLOW required the horizontal and vertical coordinates used for input files to be in meters. Starting with the release of TUFLOW 2012, the numeric engine now supports an option for customary units. We recommend that you set the units to be the desired units at the beginning of a project. SMS will allow you to switch units, but since this feature is so new, the responsibility to verify model parameters and boundary conditions are in the correct units remains with the user. Geographic Coordinates cannot be used since it is a latitude/longitude system defined in decimal degrees.

References

[1] http://www.wbmpl.com.au/
[2] http://www.tuflow.com/
[3] http://www.tuflow.com

TUFLOW Overview

TUFLOW

Overview

TUFLOW is a one-dimensional (1D) and two-dimensional (2D) flood and tide simulation software. It simulates the complex hydrodynamics of floods and tides using the full 1D St Venant equations and the full 2D free-surface shallow water equations.

TUFLOW interface

The TUFLOW interface in SMS exists in several modules including: Cartesian Grid Module, Map Module, Scatter Module, and Mesh Module. Many of the TUFLOW building blocks are managed using the Project Explorer.

How do I use TUFLOW?

The SMS tutorials are a good place to start learning to use SMS and associated models. The following tutorials will help get you started using TUFLOW.

- Models | TUFLOW 2D
- Models | TUFLOW 1D
- General | Data Visualization
- General | Observations

Numeric Engine Background

The TUFLOW computational engine computes 1D and 2D hydraulic solutions. The engine is very stable making it an excellent choice for models with lots of wetting and drying. TUFLOW includes 1D cross-sections, 1D pipe networks, 1D and 2D hydraulic structures, and allows rainfall directly on the model domain so it is useful in many different applications including coastal, riverine and urban settings.

For more information see the TUFLOW webpage www.tuflow.com [2]

Related Topics

- Define TUFLOW domain
- Mesh Module TUFLOW model
- Q&A TUFLOW
- Reading a TUFLOW simulation
- Saving TUFLOW
- TUFLOW 2D Flow constriction Shape Coverage
- TUFLOW 2D Geometry Components
- TUFLOW 2D Z Lines/Polygons (Simple) Coverage
- TUFLOW 2D Z Lines (Advanced) Coverage
- TUFLOW AD
- TUFLOW Boundary Conditions
- TUFLOW Combining 1D and 2D Domains
- TUFLOW Inlet Database
- TUFLOW Irregular Culverts
- TUFLOW Linking 2D Domains
- TUFLOW Manholes
- TUFLOW Model Parameters
- TUFLOW Network Node SX Additions
- TUFLOW Simulation
- TUFLOW Interface
- TUFLOW Check Files
- TUFLOW Grid Options
- TUFLOW Material Properties
- TUFLOW Running
- TUFLOW Reading Simulation
- TUFLOW Viewing Results

Batch Runs

The batch runs command allows multiple simulations (each with their own defined scenarios/events to run) to run simultaneously. A batch run is initiated by right-clicking on the TUFLOW simulations folder (the folder that holds the individual simulations) and choosing “Batch Run.”

The batch run command brings up a dialog to choose the simulations to run. After checking the simulations to run, you can choose whether or to “Export and Run” the simulations or “Run only.” The run only option can be used if the simulation files have been exported previously and don’t need to be changed.

After the simulations have been selected in the batch run dialog, the runs will be launched and can be monitored in the “Model wrapper” dialog. The top part of this dialog has progress bars showing the individual progress of each run. The screen output for each run can be viewed by clicking on the associated progress bar. The screen output is shown in the bottom part of the dialog.

Command Objects

Command Objects provide the ability to provide additional commands for TUFLOW simulations using the TUFLOW command file syntax. The SMS interface provides some guidance for using these commands and allows users to link to objects created in SMS. Command objects can be used for:

- Create scenarios to do multiple runs using the same simulation.
- Share commands between multiple simulations or 2D Geometry components.
- Utilize commands not available in the SMS interface.
- Enter commands to read GIS data generated outside of SMS.

TUFLOW Command Object syntax is described in the TUFLOW Manual. Each line (except for blank/comment lines) has a command identifier. After the identifier most commands have are followed by a double equals “==” and a command value. In some cases, the command value may have multiple parts separated by blank spaces or “|.” A few commands don’t have any arguments and do not have a “==” at all.

Types of Commands

There are several kinds of commands that can be utilized through command objects. The types of commands that can be issued used include: 2D Model control commands (*.tcf), 1D Model control commands (ecf), 2D Geometry commands (tgc), Boundary conditions commands (*.tbc), and commands for specific events (*.tef). The event commands are defined with each event when using composite events. The rest are managed in the project explorer as command objects.

The Appendix of the TUFLOW manual contains a complete list of the commands that are available, the purpose of each command, and its syntax. The identifiers above in parentheses (*.tcf, *.ecf, *.tgc, *.tbc, and *.tef) are the abbreviations used by TUFLOW for file naming and also used to find the appropriate section of the Appendix for a set of commands.

Creating Command Objects

Command objects are created using right-click commands in the project explorer. After creating a TUFLOW simulation, you should see a folder named “Command Objects” in the TUFLOW root folder. You can create command objects by right clicking on the “Command Objects” folder, choosing “Create Command Object”, and choosing the appropriate type. This will create a folder for command objects of this type along with a new command object. This approach can be repeated to create additional command objects of the same or different type.

Editing Command Object Contents

When command objects are first created, their contents are blank. You can edit the contents of a command object by double clicking on the object or right-clicking and choosing “edit.” This will bring up the command object editor dialog.

The command object editor dialog has two main parts. One part consists only of a text editor. The text in this editor is what is stored with the command object and will be used with the TUFLOW model. The commands for the object may be typed directly into the editor. Comments may be added and must start with the exclamation mark (!).

The other part of the dialog contains controls to assist the user in creating commands for the command object. There is a control showing the list of commands in a tree structure organized by the appendix where the command is found in the TUFLOW manual. This part of the dialog will also contain other controls based upon the command that is selected in tree control. For example, if the command requires a single floating point value a edit control will shown where the user can specify the value for the command.

Once the command creation controls have been specified as desired, the command is inserted into the command text using the insert button. Once this has been done, editing of the command must be done in the editor. You can’t edit commands already created using the create command controls. You can delete an existing command and insert another if you wish to use the controls to setup the command text.

Using Command Objects in Simulations

With the exception of event command objects (discussed later), command objects are used by creating links to the objects from simulations and geometry components. Links are created by dragging the command object under the object you wish to add the link to. 2d model control command objects and 1d model control command objects are linked to simulations. 2D geometry commands and boundary condition commands are linked to 2D Geometry Components.

When a command object is linked to a simulation or geometry component, SMS writes a file with the exact text from the command object and links to the file from the TCF or TGC file as appropriate. Command objects are always used after any commands specified in the SMS interface and therefore will override duplicate specifications. Multiple command objects can be linked to the same object and will be used in order from top to bottom. Command objects can override settings defined in command objects used previously.

Event command objects are only used with composite events (** need to add link to this help topic). To set commands for a composite event, you click on the button “Additional commands” in the TUFLOW events dialog (***link to dialog help). This text will be stored with the event and written to a file with the extension “tef” which will be used when running TUFLOW. The event command objects do not show up in the project explorer.

Scenarios in Command Objects

TUFLOW command objects can define scenarios in order to setup multiple models that have shared data without the need for separate simulation objects. Anything not specified for one or more scenarios is used in all scenarios for the simulation. Scenarios could be used to model different geometries by including z-lines or tins for specific scenarios but not others.

Scenarios can only be defined using command objects. Scenarios are defined by using the following commands:

- If Scenario ==
- Else If Scenario ==
- Else
- End If

A scenario block must start with “If Scenario” and end with “End If.” The other commands are optional.

Define TUFLOW domain

TUFLOW supports 2D domains, 1D cross-section based domains, and 1D pipe network domains. Multiple domains of each type can be used within the same TUFLOW simulation. To use Multiple 2D domains in TUFLOW, you must purchase the add-on module.

Types of Domains

2D Domains

Two-dimensional domains are Cartesian grid (rectilinear) and include elevation data.

TUFLOW 2D domains are cartesian (rectilinear) grids. 2D domains are created using a Cartesian Grid Frame inside a TUFLOW Grid Extents Coverage.

Grids store elevation values at each corner, midside, and center. There are several Grid Options that apply to a grid that can be specified by right-clicking on the grid and choosing *Properties*.

A 2D domain is represented as a 2D Geometry Component in the TUFLOW interface. A 2D Geometry component includes the grid and associated coverages.

Grid elevations can be modified using the select grid location tool by selecting and changing the z values at specific directions. A TUFLOW 2D Z Lines/Polygons (Simple) coverage can also be used to force elevations using arcs or polygons.

Multiple grids can be used inside the same simulation. The grid used in each area of the domain is specified using a TUFLOW 2D/2D Link Coverage.

1D Cross-section Domains

Cross-section based domains include channel and cross-section definitions.

1D cross-section domains use both the TUFLOW network coverage and the TUFLOW cross-section coverage. The network coverage describes the channel attributes such as the length and the cross-sections are defined in the cross-section coverage.

1D Pipe Network Domains

Pipe network domains can be used to model such things as culverts and storm sewers.

1D Pipe networks are created using culvert channels inside a 1D Network Coverage. The shape of the pipe segments are specified using the attributes button in the channel attributes dialog.

Channel inlets, storage, and invert elevations can be defined on the arc nodes (endpoints). Starting in version 10.1, you can create your own inlet definitions and use them within your pipe network model. See TUFLOW_Inlet_Database

Combining Multiple Domains

1D to 2D Domains

A very powerful and useful feature in TUFLOW is the ability to combine a 1D cross-section domain and a 2D domain. This method can be used to embed a 2D domain between two 1D domains, embedding pipe networks, or to model the channel in 1D and floodplain in 2D. See Combining 1D and 2D Domains.

2D Domain to 2D Domain

One of the disadvantages to TUFLOW is that it uses a fixed grid rather than a mesh which can transition very well from large to small cells. This can be overcome somewhat by using 1D and 2D domains. TUFLOW also supports using multiple 2D domains of different resolutions. The SMS interface makes it easy to set up see Linking 2D Domains.

Q&A TUFLOW

Return to the Main Q&A Page

Q: When running TUFLOW, nothing gets calculated?

A: This can happen if you do not have a driver installed for TUFLOW. To download the driver go to this link http://www.tuflow.com/Downloads_Misc.htm and click on the driver download.

Reading a TUFLOW Simulation

Reading a TUFLOW simulation in SMS

TUFLOW simulation files can be opened into SMS by opening the simulation's control file (*.tcf). This can be done by dragging and dropping the file into the graphics window or by using File | Open. Multiple simulations can be opened together, and SMS will only create one copy of the data that is shared between the projects.

TCF File

The following commands are supported when reading a TCF file into SMS:

- BC CONTROL FILE – read the specified TUFLOW Boundary Condition file
- BC DATABASE – set the active database
- BC EVENT NAME – set the active event filename to be used
- BC EVENT TEXT – set the text to be substituted by the BC EVENT NAME value
- CELL SIDE WET/DRY DEPTH – set the depth for when cell sides wet and dry
- CELL WET/DRY DEPTH – set the wet/dry depth for when a cell wets and dries
- CHECK INSIDE GRID – set to ERROR, WARNING, or OFF
- DISPLAY WATER LEVEL – display water level for cell at specified location
- END 1D DOMAIN – end block of 1D commands
- END 2D DOMAIN – end block of 2D domain definition
- END TIME – set the simulation finish time in hours
- ESTRY CONTROL FILE (AUTO) – set the ESTRY control filename
- GEOMETRY CONTROL FILE – read the specified TUFLOW Geometry Control file
- HX ZC CHECK – set whether or not to check the minimum ZC elevation is above the 1D bed level
- INSTABILITY WATER LEVEL – set the water level used to detect instabilities
- MAP OUTPUT DATA TYPES – define data types to be output in map format
- MAP OUTPUT INTERVAL – set the output interval for map based output in seconds
- MASS BALANCE OUTPUT – set whether or not to output mass balance data
- MI PROJECTION – set the geographic projection
- NUMBER ITERATIONS – specify the number of iterations per timestep
- READ FILE – read the specified external file as an addition TUFLOW control file
- READ GIS FC – read the specified flow constriction mif/mid files
- READ GIS IWL – read the specified initial water level mif/mid files
- READ MATERIALS FILE – read the specified TUFLOW materials file
- READ MI PROJECTION - set the projection
- READ RESTART FILE – set the model to use the specified restart file

- SCREEN/LOG DISPLAY INTERVAL – set the frequency for display of output to the computer screen and log file
- SET IWL – set a default initial water level for all cells
- START 1D DOMAIN – start a block of 1D commands
- START 2D DOMAIN – begin block of 2D domain definition
- START MAP OUTPUT – set simulation time in hours when map output begins
- START TIME – set the start time of the simulation in hours
- STORE MAXIMUMS AND MINIMUMS – set whether or not to store the highest and lowest values
- SX ZC CHECK – set whether or not to check if the minimum ZC elevation is below the connected or snapped 1D node bed level
- TIMESTEP – set the computation time step in seconds
- VISCOSITY COEFFICIENT – set the viscosity coefficient(s)
- VISCOSITY FORMULATIONS – set the viscosity formulation
- WRITE RESTART FILE AT TIME – set when to write the restart file in hours
- WRITE RESTART FILE INTERVAL – set the interval in hours between writing the restart file
- ZERO NEGATIVE DEPTHS IN SMS – set whether or not to zero depths if negative

The following commands are not supported and are written to an external file:

- ADJUST HEAD AT ESTRY INTERFACE
- APPLY WAVE RADIATION STRESSES
- APPLY WIND STRESSES
- BC WET/DRY METHOD
- BC ZERO FLOW
- BED RESISTANCE CELL SIDES
- BED RESISTANCE DEPTH INTERPOLATION
- BED RESISTANCE VALUES
- BOUNDARY CELL SELECTION
- CALIBRATION POINTS MI FILE
- CHANGE ZERO MATERIAL VALUES TO ONE
- CHECK MI SAVE EXT
- CSV TIME
- DEFAULTS
- DENSITY OF AIR
- DENSITY OF WATER
- DEPTH/RIPPLE HEIGHT FACTOR LIMIT
- DISTRIBUTE HX FLOWS
- DOUBLE PRECISION
- EXCEL START DATE
- FIRST SWEEP DIRECTION
- FREE OVERFALL
- FREE OVERFALL FACTOR
- FROUDE CHECK
- FROUDE DEPTH ADJUSTMENT
- GLOBAL FC CH FACTOR
- GLOBAL WEIR FACTOR
- INPUT DRIVE
- INSIDE REGION
- LATITUDE

- LINE CELL SELECTION
- MAP CUTOFF DEPTH
- MAXIMUM POINTS
- MAXIMUM VERTICES
- MESHPARTS
- MODEL PRECISION
- NULL CELL CHECKS
- NUMBER 2D2D LINK ITERATIONS
- OBLIQUE BOUNDARY ALIGNMENT
- OBLIQUE BOUNDARY METHOD
- OUTPUT DRIVE
- READ GIS CYCLONE
- READ GIS HURRICANE
- READ GIS ISIS NETWORK
- READ GIS ISIS NODES
- READ GIS ISIS WLL
- READ GIS ISIS WLL POINTS
- READ GIS GLO
- READ GISLP
- READ GISPO
- READ GIS XP NETWORK
- READ GIS XP NODES
- READ GISXP WLL
- READ GIS XP WLL POINTS
- READ ROWCOL IWL
- RECALCULATE CHEZY INTERVAL
- SHALLOW DEPTH STABILITY FACTOR
- SHALLOW DEPTH WEIR FACTOR CUT OFF DEPTH
- SHALLOW DEPTH WEIR FACTOR MULTIPLIER
- SNAP TOLERANCE
- START TIME SERIES OUTPUT
- START WIND OUTPUT AT TIME
- SUPERCRITICAL
- SX HEAD ADJUSTMENT
- TIME SERIES OUTPUT INTERVAL
- TIMESTEP DURING WARMUP
- UK HAZARD DEBRIS FACTOR
- UZ HAZARD FORMULA
- UZ HAZARD LAND USE
- UNUSED HX AND SX CONNECTIONS
- VERBOSE
- VG Z ADJUSTMENT
- WARMUP TIME
- WATER LEVEL CHECKS
- WAVE PERIOD
- WETTING AND DRYING
- WIND OUTPUT INTERVAL

- WIND/WAVE SHALLOW DEPTHS
- WRITE EMPTY MI FILES
- WRITE PO ONLINE
- WRITE Z1D CHECK FILES

The following commands are ignored when read in by SMS because they are hardwired:

- CHECK MI SAVE DATE – hardwired to “OFF”
- LOG FOLDER – hardwired to “.\\log”
- MI PROJECTION CHECK – hardwired to “WARNING”
- OUTPUT FOLDER – hardwired to “..\\results”
- WRITE CHECK FILES – hardwired to “..\\check\\”

TGC File

The following commands are supported when reading a TGC file into SMS:

- ALLOW DANGLING Z LINES – allows dangling z lines
- CELL SIZE – sets the cell size
- GRID SIZE (N, M) – sets the number of I and J in the grid
- GRID SIZE (X, Y) – sets the grid size in x and y
- ORIENTATION –- sets the x,y location of a point along the x-axis
- ORIENTATION ANGLE – sets the angle of the grid
- ORIGIN – sets the location for the origin
- READ FILE – reads an external file
- READ GIS CODE – reads mif/mid files and stores the codes in polygons in a bc coverage
- READ GIS FC SHAPE – reads mif/mid files to create a flow constriction coverage
- READ GIS FLC – reads mif/mid files to create a 2D spatial coverage with form loss coefficient as the coverage property
- READ GIS IWL – reads mif/mid files to create a 2D spatial coverage with initial water level as the coverage property
- READ GIS LAYERED FC SHAPE – reads mif/mid files to create a flow constriction coverage
- READ GIS LOCATION – set the grid location
- READ GIS MAT – reads mif/mid files to create an area property coverage
- READ GIS VARIABLE Z SHAPE – reads mif/mid files to create a z-shape coverage
- READ GIS WRF – reads mif/mid files to create a 2D spatial coverage with weir factor as the coverage property
- READ GIS Z LINE – reads mif/mid files to create a z-lines coverage
- READ GIS Z SHAPE – reads mif/mid files to create a z-shape coverage
- READ GIS ZPTS – reads mif/mid files to create points in a z-lines coverage
- READ ROWCOL CODE – sets code values for grid cells
- READ ROWCOL MAT – sets material values for grid cells
- READ ROWCOL ZPTS – sets z values for grid cells
- READ TIN ZPTS – creates a scatter set from the points
- SET CODE – set the default code
- SET IWL – if opening with a simulation, sets the default initial water level, otherwise the card is written to an external file
- SET MAT – set the default material id
- SET ZPT – set the default z-value for each grid cell

The following commands are not supported and are written to an external file:

- READ TGA

- READ TGC
- EXTERNAL BNDY
- READ GIS CNM
- READ GIS FRIC
- READ ROWCOL GRID
- SET CNM
- SET FRIC
- SET CODE ZERO ABOVE ZC
- WRITE MI DOMAIN
- WRITE MI GRID
- CREATE TIN ZPTS
- DEFAULT LAND Z
- INTERPOLATE ZC
- INTERPOLATE ZHC
- INTERPOLATE ZUV
- INTERPOLATE ZUVC
- INTERPOLATE ZUVH
- PAUSE WHEN POLYLINE DOES NOT FIND ZPT
- TIN COINCIDENT POINT DISTANCE
- READ MI Z HX LINE
- READ MI Z HX LINE RIDGE or MAX or RAISE
- READ MI Z LINE RIDGE or MAX | GULLY or MIN | HX
- READ MI Z SHAPE RIDGE or MAX or RAISE | GULLY or MIN or LOWER

TBC File

The following commands are supported when reading a TBC file into SMS:

- BC DATABASE – set the database to be used
- BC EVENT NAME – set the name of the event file to be used
- BC EVENT TEXT – set the text to be replaced by the event name
- READ GIS BC – read the boundary condition MIF/MID files
- READ GIS RF – read the rainfall MIF/MID files
- READ GIS SA – read the source flow MIF/MID files

The following commands are not supported and are written to an external file:

- GLOBAL RAINFALL AREA FACTOR
- GLOBAL RAINFALL BC
- GLOBAL RAINFALL CONTINUIN LOSS
- GLOBAL RAINFALL INITIAL LOSS
- READ GIS SA ALL
- READ GIS SA PITS
- READ GIS RF
- READ ROWCOL RF
- UNUSED HX AND SX CONNECTIONS

ECF File

The following commands are supported when reading a ECF file into SMS:

- DEPTH LIMIT FACTOR – set the depth limit for detecting instabilities
- MANHOLE DEFAULT C EXIT COEFFICIENT – set the K coefficient for automatic circular manholes
- MANHOLE DEFAULT LOSS APPROACH – set the loss approach for automatic manholes
- MANHOLE DEFAULT R EXIT COEFFICIENT – set the K coefficient for automatic rectangular manholes
- MANHOLE DEFAULT SIDE CLEARANCE – set the side clearance for circular and rectangular manholes (automatic)
- MANHOLE DEFAULT TYPE – set the type (circular, rectangular, no chamber, automatic) for automatic manholes
- MANHOLE K MAXIMUM BEND/DROP – set the K coefficient for energy loss when using Engelhund approach with automatic manholes
- MANHOLE MINIMUM DIMENSION – set the minimum dimension for circular and rectangular automatic manholes
- MANHOLES AT ALL CULVERT JUNCTIONS – set whether or not to automatically create manholes at culvert junctions
- MINIMUM NA – set the minimum surface area in all NA tables
- OUTPUT INTERVAL – set the output interval for ESTRY output
- PIT INLET DATABASE – read the specified inlet database file
- READ GIS NETWORK – read the network mif/mid files
- READ GIS TABLE LINKS – read the table link mif/mid files
- READ GISI WLL – read water level lines mif/mid files
- READ GIS WLL POINTS – read elevations and points from water level lines from mif/mid files
- STORAGE ABOVE STRUCTURE OBVERT – define how surface area is to be contributed to the NA table
- TIMESTEP – set the computation time step in seconds
- READ FILE – read the external file as an ECF file
- SET IWL – set the initial water level

The following commands are not supported and are written to an external file:

- BG DATA
- BRIDGE FLOW
- CHECK MI SAVE DATE
- CHECK MI SAVE EXT
- CREATE NODES
- CS DATA
- CSV FORMAT
- CSV TIME
- CULVERT ADD DYNAMIC HEAD
- CULVERT CRITICAL H/D
- CULVERT FLOW
- DEFAULTS
- END TIME
- FIXED FIELD FLAGS
- FLOW CALCULATION
- FROUDE CHECK
- HEAD RATE CREEP FACTOR
- HEAD RATE LIMIT

- HEAD RATE LIMIT MINIMUM
- INTERPOLATE CROSS-SECTIONS
- INTERPOLATE CULVERT INVERTS
- LOG FOLDER
- M11 NETWORK
- MI PROJECTION
- MINIMUM CHANNEL STORAGE LENGTH
- MINIMUM NA PIT
- MOMENTUM EQUATION
- NA DATA
- ORDER OUTPUT
- OUTPUT FOLDER
- OUTPUT TIMES SAME AS 2D
- PIT CHANNEL OFFSET
- READ MATERIALS FILE
- READ GIS IWL
- RELATIVE RESISTANCE
- S CHANNEL APPROACH
- SNAP TOLERANCE
- START OUTPUT
- START TIME
- STRUCTURE LOSSES
- TAPER CLOSED NA TABLE
- TRIM XZ PROFILES
- VEL RATE CREEP FACTOR
- VEL RATE LIMIT
- VEL RATE LIMIT MINIMUM
- VG DATA
- WEIR FLOW
- WLL ADDITIONAL POINTS
- WLL ADJUST XS WIDTH
- WLL AUTOMATIC
- WLL NO WEIRS
- WLL VERTICAL OFFSET
- WLLP INTERPOLATE BED
- WRITE CHECK FILES
- WRITE CSV ONLINE
- WRITE EMPTY MI FILES
- XS DATABASE
- ZERO CULVERT COEFFICIENTS

The following commands are ignored when read in by SMS because they are hardwired:

- APPLY ALL INVERTS – hardwired to “ON”
- CONVEYANCE CALCULATION – hardwired to “ALL PARALLEL”
- FLOW AREA – hardwired to “TOTAL”
- WLL APPROACH – hardwired to “Method B”

Saving TUFLOW

When you do a *File | Save As...* the following files get saved in the *.sms

- *.mat referenced to new save location
- _grds.h5 referenced to new save location
- *.mat_h5 referenced to new save location
- Geomcomps.h5 referenced to new save location
- scatter.h5 referenced to new save location
- tuflow files don't get saved to new location unless exported again

TUFLOW 2D Geometry Components

A 2D geometry component groups grids and data that apply to the grid. The other kinds of data in a geometry component include coverages and TIN (scatterset). A TIN is used to modify the Z values. The purpose of each coverage is dependent upon the coverage type. Some coverages will modify z values and others will control other attributes. There can be multiple objects of the same type in a geometry component. When this happens, the data is applied in the order of the objects in the component. This means that the objects below in the project explorer are the ones that will control. This applies to Z as well as other attributes. Links can be rearranged by dragging them up or down within the 2D Geometry domain.

Coverages in a geometry component

TUFLOW 2D Geometry components associate grids with TUFLOW coverages that apply specifically to the domain. For example a grid component may contain an area property coverage that modifies the materials in the grid. Grids and coverages are represented as TUFLOW Project Explorer Links inside of a 2D Geometry component. Only one grid may be linked in each geometry component.

The following coverages can be put inside of a 2D Geometry domain

- Area Property Coverage
- Boundary Condition Coverage
- 2D Z Lines/Polygons (Simple)
- 2D Spatial Attributes Coverage
- 2D Z Lines (Advanced)

TINs (scatterset) in a geometry component

- In addition, Scatter Data can also be put inside a 2D Geometry domain to modify Z values. Unlike the other components, it has additional TUFLOW options that can be set in its link's right-click menu. The TUFLOW Options that can be set for how the scatter data modifies Z values are All, Add, Min and Max.
- Clip Regions for Read Tin z pts

Considerations for modifying Z data

As mentioned previously, the order links are listed matters for coverages and scatter data. This can be particularly useful for Z data but you have make sure you are applying the attributes in the correct order.

Suppose you have a situation where you want to incorporate a 2.0 m high fenceline and also raise a portion of the domain to model a proposed fill for a development. The fenceline is defined a z-line and the raised domain is a TIN (you could use a 2D z lines/polygons (simple) if it is a simple fill scenario). If the fenceline is above the TIN in the project explorer, anywhere that they overlap the TIN would take precedence and the fenceline would get wiped out. If the fenceline is below the TIN, the TIN elevations will get applied and then the fenceline will raise the elevations 2.0 m above the new elevations. Depending upon the scenario you want to represent, either option may be the preferred mechanism.

External files

- An external file can be read in as part of a geometry component. As previously mentioned, the order of links is important, and the external file behaves the same way as the z-modifying data. An external file is added by right-clicking on the geometry component and selecting "Add External File...". After a file has been selected, a tree item with the filename will appear at the bottom of the geometry component tree item group. This external file item can be moved around to the desired location and the filename can be changed by right-clicking and selecting "Browse...".

TUFLOW AD

SMS now supports the TUFLOW Advection Dispersion (AD) module. This allows users to model constituents using TUFLOW.

Creating the Constituents

The constituent data is entered in the Model Control of the TUFLOW simulation. The Constituents tab allows the user to enter in the following parameters for each constituent:

- Name
- Decay rate (units of day^-1)
- Settling rate (units of m/day)
- Longitudinal Dispersion Coefficient
- Transverse Dispersion Coefficient
- Initial Condition – Specified Value or Polygon by Polygon value
- Minimum Dispersion – Specified Value or Polygon by Polygon value

Boundary Conditions

For each boundary condition and constituent the user can provide a curve representing concentration at the boundary. This can be done by selecting the Advection Dispersion BC... button in the properties dialog for the arc or polygon.

TUFLOW Boundary Conditions

TUFLOW Boundary conditions may be defined at nodes, arcs, or polygons in a Boundary Conditions Coverage.

Boundary Condition Locations

1D Boundary Conditions at Nodes

Boundary conditions defined at points are 1D boundary conditions and must be placed at the same location as a 1D boundary node. (See Snapping Feature Objects) The boundary condition information is specified in the BC Node Attributes Dialog.

2D Boundary Conditions at Arcs

Arcs can be used to define 2D Boundary conditions applied to the 2D domain. The BC attributes are specified in the BC Arc Attributes Dialog.

2D Boundary Conditions for Polygons

Polygons can be used to define rainfall applied to 2D domains or to specify active/inactive information for 2D domains. The BC Polygon Attributes dialog specifies the type of information stored with the polygons.

Convert GIS Rainfall Data to TUFLOW BC Polygons

GIS files containing TUFLOW rainfall data may be imported into SMS and a Boundary Condition coverage with the data assigned to polygons will be created. The data for each polygon should be contained in the following columns: 8 STARTYEAR – year of first data point (integer)

- STARTMONTH – month of first data point (integer)
- STARTDAY – day of first data point (integer)
- STARTHOUR – hour of first data point (integer)
- STARTMIN – minutes of first data point (integer)
- STARTSEC – seconds of first data point (integer)
- TIMESPAN – timespan between data point in minutes (integer)
- NUMRAINPTS – number of data points (integer)
- PT0 – data for the first point (double)
- PT1, PT2,... – data for each point

When converting GIS rainfall data to TUFLOW rainfall polygons, the user will be prompted to set a reference time. Each rainfall data point will be stored as total mm for each interval of time (hrs) from the reference time.

Events

Every boundary condition should be setup for each event that will be used with the boundary condition. The events can be defined by clicking on the Add/Remove Events button. The list of events is common to all the boundary conditions in the project.

Options

Depending upon the type of boundary condition, there are several additional boundary condition options.

- Override Component Identifier – SMS creates a default unique boundary condition identifier based upon the coverage and id of the arc. You can override this value to more easily find boundary condition information in the TUFLOW check, and log files.
- Spline curve – If this option is on, TUFLOW will smooth the boundary condition curve using a spline algorithm. This should generally only be used for smooth, cyclic boundary conditions such as a tidal boundary.

Boundary Condition Types

The types of boundary conditions available depend upon whether the boundary condition is at nodes, arcs, or polygons.

Boundary Conditions'

Type	Available Locations	Description
Not used	Points, Arcs	No boundary condition exists at the location.
Flow vs Time	Points, Arcs	Sets the flowrate as a curve through time.
Flow vs Wse	Points	Sets the flowrate based upon a relationship with the water surface elevation.
Wse vs Time	Points, Arcs	Sets the water surface elevation as a curve through time.
Wse vs Flow	Points, Arcs	Defines a relationship to determine the water surface elevation from a flowrate. The relationship can be specified or TUFLOW can compute it using water surface slope.
1D Flow/2D Water Level Connection (HX)	Arcs	Defines the interface between a 1D and 2D domain. See Combining 1D and 2D Domains.
Non-directional Flow vs Time	Arcs	Defines a flowrate that enters along each cell of the boundary.
Non-directional Flow vs Wse	Arcs	Defines a flowrate that enters along each cell of the boundary based upon a relationship to the water surface elevation.
Flow Source from 1D model (SX)	Arcs	Defines an interface between a 1D and 2D domain which is often useful for curlverts rather than using the 1D Flow/2D Water Level Connection.
Flow/Rainfall over Area	Polygons	Defines either an amount of flow entering the polygon (applies to all wet cells or lowest cell if none) or rainfall amounts (applies to all cells in the polygon).
Cell Codes	Polygons	Specifies whether cells are active or inactive and whether or not they will be included in the mesh output files.

TUFLOW Check Files

TUFLOW provides excellent feedback for finding and fixing model errors. This feedback exists in several files. All of these files are prefixed by the name of the simulation that you are running and the filename ending identifies the type of data contained in the file.

Log Files

In the \runs\log directory (within the TUFLOW files for your simulation), are the log files for individual run.

The most useful file for finding model problems is the _messages.mif file. This file contains spatially located messages including errors, warnings or checks (listed in descending order of severity). These can be read into the GIS module of SMS. This provides quick feedback of model errors and the exact locations that the errors occur. This is very useful for pinpointing model setup problems and should be the first step in fixing model issues.

The *.tlf file contains information (in ASCII format) about how the run proceeded. It includes information on the parameters and settings used, the files that were read and used, as well as a copy of the screen information that is written as TUFLOW runs. This file can be used to verify the model is setup and is running as expected.

Check Files

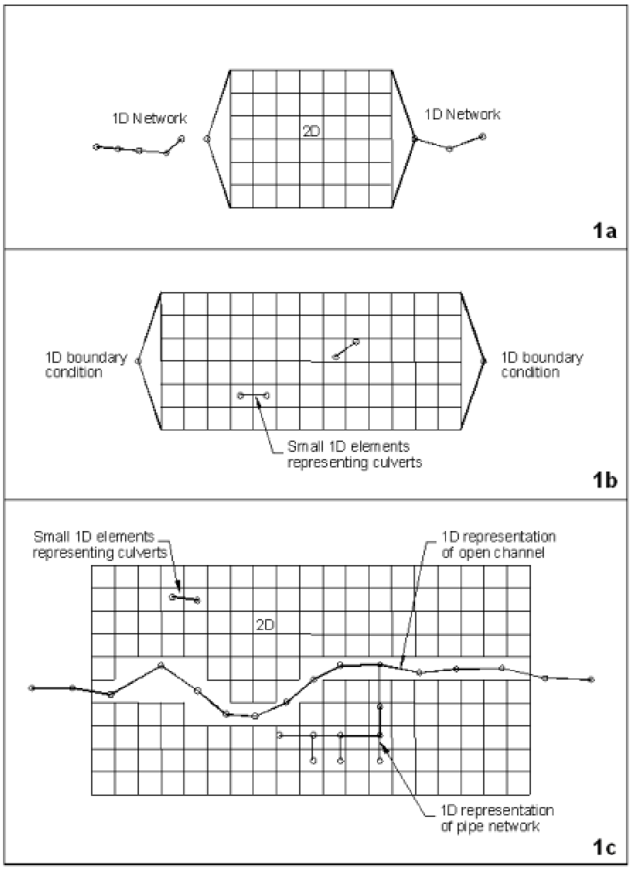
In addition to the log files mentioned previously, TUFLOW writes additional files that can be used to verify model setup. These files are in the directory labeled "\\check" and are only written if the 'Write Check Files' toggle is checked in the misc tab in the simulation model control dialog (this is the default). Many of the check files are in MapInfo MIF/MID files that can be read into the SMS GIS module where the info tool can be used to see the values at individual locations.

TUFLOW generates potentially lots and lots of check files. A description of the check files and what they contain can be found in the TUFLOW manual.

TUFLOW Combining 1D and 2D Domains

1D and 2D Domains can be linked in three different ways.

1. A 2D domain within a 1D domain (1a)
2. 1D elements representing culverts or pipe networks (1b or 1c)
3. 1D cross-section based domain for river channel and 2D domain for floodplain (1c)



Linking 1D Pipe network within a 2D network

The easiest way to link a 1D pipe network to a 2D network is to double click on one of the nodes in the network coverage and define an inlet (connection to the 2D domain). The inlet can be circular or rectangular and the shape and losses can be specified.

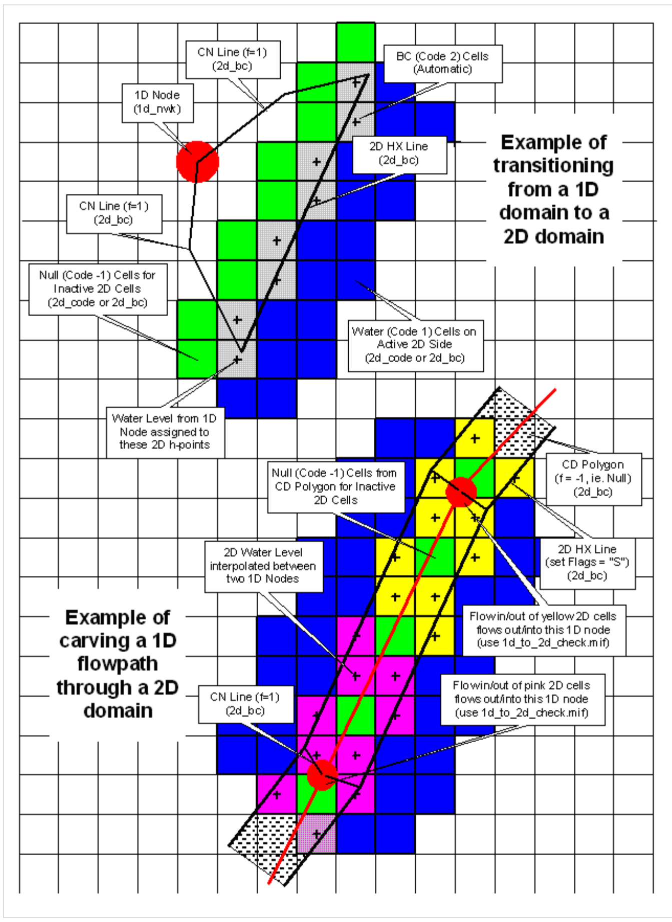
Linking 1D domain and 2D domains

Except for pipe networks defined above, linking a 1D network to a 2D network requires several coverages.

- Network coverage – contains the 1D channels
- BC coverage – 1D Flow/2D Water Level Connection (HX) or Flow source from a 1D model (SX) arcs define the interface between the 1D and 2D domains.
- 1D/2D Connections – Arcs connect 1D nodes to ends of 1D Flow/2D Water Level arcs or Flow source arcs. When used with Flow source arcs only 1 connection arc is needed where the 1D Flow/2D Water Level arcs require 2 arcs (one to each end of the arc).

Arcs in the connections coverage must snap to the 1D network nodes and the endpoints of the 1D Flow/2D Water Level Connection arcs. The snapping option setup in the map tab of the preferences dialog should be used to ensure that the nodes are placed correctly.

The following image shows how 1D/2D linkages are setup both for both items 1 and 3 above.


CN Line (f=1)
(2d_bc)
BC (Code 2) Cells
(Automatic)
1D Node
(1d_nwk)
2D HX Line
(2d_bc)
Example of transitioning from a 1D domain to a 2D domain
CN Line (f=1)
(2d_bc)
Null (Code -1) Cells for Inactive 2D Cells
(2d_code or 2d_bc)
Water (Code 1) Cells on Active 2D Side
(2d_code or 2d_bc)
Water Level from 1D Node assigned to these 2D h-points
Null (Code -1) Cells from CD Polygon for Inactive 2D Cells
CD Polygon
(f = -1, ie. Null)
(2d_bc)
2D Water Level interpolated between two 1D Nodes
2D HX Line
(set Flags = "S")
(2d_bc)
Flowin/out of yellow 2D cells flows out/into this 1D node
(use 1d_to_2d_check.mif)
Example of carving a 1D flowpath through a 2D domain
CN Line (f=1)
(2d_bc)
Flowin/out of pink 2D cells flows out/into this 1D node
(use 1d_to_2d_check.mif)

TUFLOW Grid Options

The TUFLOW Grid Options dialog is used to specify how TUFLOW will handle the grid data. Some of the options deal with what types of data will be specified on the grid cells. This data can alternatively be specified using coverages and this has several advantages including size of the data, speed of execution, and more opportunities for reuse.

The general options include:

• Allowing dangling z lines – This turns on/off the TUFLOW command to check for dialing z lines (see the TUFLOW manual for description).

• TUFLOW External Command Files – This can be used to read an external file from the TUFLOW *.tgc file.

• Time step – Since the required time step depends heavily on the size of grid cells, it is often convenient for each grid to have a unique timestep when using multiple grids in a simulation. These controls allow the user to override the global timestep on a grid by grid basis.

The rest of the dialog is used to tell SMS what kind of data to write from the grid in SMS. The optional data types are materials and cell codes. Materials may be specified cell by cell in the grid or by using an area property coverage. If using an area property coverage, a default material is specified for any areas not inside of a polygon. Likewise cell codes can be specified cell by cell, or from polygons in a TUFLOW BC Coverage.

TUFLOW Inlet Database

TUFLOW supports several types of inlets that connect 1D pipe network domains to 2D overland domains. These include circular, rectangular, and weir. TUFLOW now also supports the ability to define depth vs flow curves to allow for more flexibility. The user defined inlet information is stored in an inlet database.

The inlet database stores information about the user defined inlet types. This information includes the flow area, width, and the depth vs flow curve. The flow area is used to compute the velocities in the inlet channel and the width is used to determine how many 2D cells to connect the inlet to. For more information about how TUFLOW uses the inlet database, see the TUFLOW documentation.

To define the Inlet Database

- Create a 1D network coverage and an arc.
- Double click on the end node of an arc (which is where inlets are defined)
- Change the inlet type to "Inlet — From Database"
- Click on the button labeled "Inlet Database..."
- In the dialog that comes up, define the type (name), flow area, flow width, and curve for each of the inlets you wish to define.

To use an inlet from an Inlet Database

- Create a 1D network coverage and an arc.
- Double click on the end node of an arc (which is where inlets are defined)
- Change the inlet type to "Inlet – From Database"
- Select the inlet type by name in the combo-box labeled "Inlet Database Name."

Importing/Exporting the Inlet Database

From the inlet database dialog, it is possible to export or import the database in to the format used by TUFLOW (two or more csv files). This format is very easy to create or use for graphs using a spreadsheet program. For more information on this format, see the TUFLOW documentation.

TUFLOW Interface

The TUFLOW interface in SMS exists in several modules including: Cartesian Grid Module, Map Module, Scatter Module, and Mesh Module. Many of the TUFLOW building blocks are managed using the Project Explorer.

Tutorials

The SMS tutorials are a good place to start learning to use SMS and associated models. The following tutorials will help get you started using TUFLOW.

- Models | TUFLOW 2D
- Models | TUFLOW 1D
- General | Data Visualization
- General | Observation

General Steps to Build a TUFLOW Model

- Define the domain or domains
- Setup boundary conditions
- Create the simulation
- Define Material Properties
- Set Model Parameters
- Run TUFLOW
- Find problems and verify model setup using check files
- View and analyze the results
- Repeat above steps to generate new simulations

Optional Steps

- Define flow constrictions
- Define geometry modifications such as levees

TUFLOW Coverages

Much of the data used in TUFLOW is specified on a coverage level into one of the TUFLOW Coverages.

TUFLOW Irregular Culverts

SMS now supports irregular culverts for TUFLOW.

Creating the Irregular Culverts

In the network coverage, change the type of an arc to "Irregular Culvert". By clicking on "Attributes", the user can specify what shape the culvert is. The drop-down specifies what polygon will be used for this arc. New polygons can be created and named. The table specifies the points of the polygon in clock-wise order. Please note that negative values cannot be entered for y-values. The number of barrels, contraction coefficients, and losses can be entered in this dialog, similar to other culverts. The perimeter length of the polygon, and its area, are given for convenience.

When the TUFLOW simulation is exported, an HW table will be generated for each unique irregular culvert shape in use.

TUFLOW Linking 2D Domains

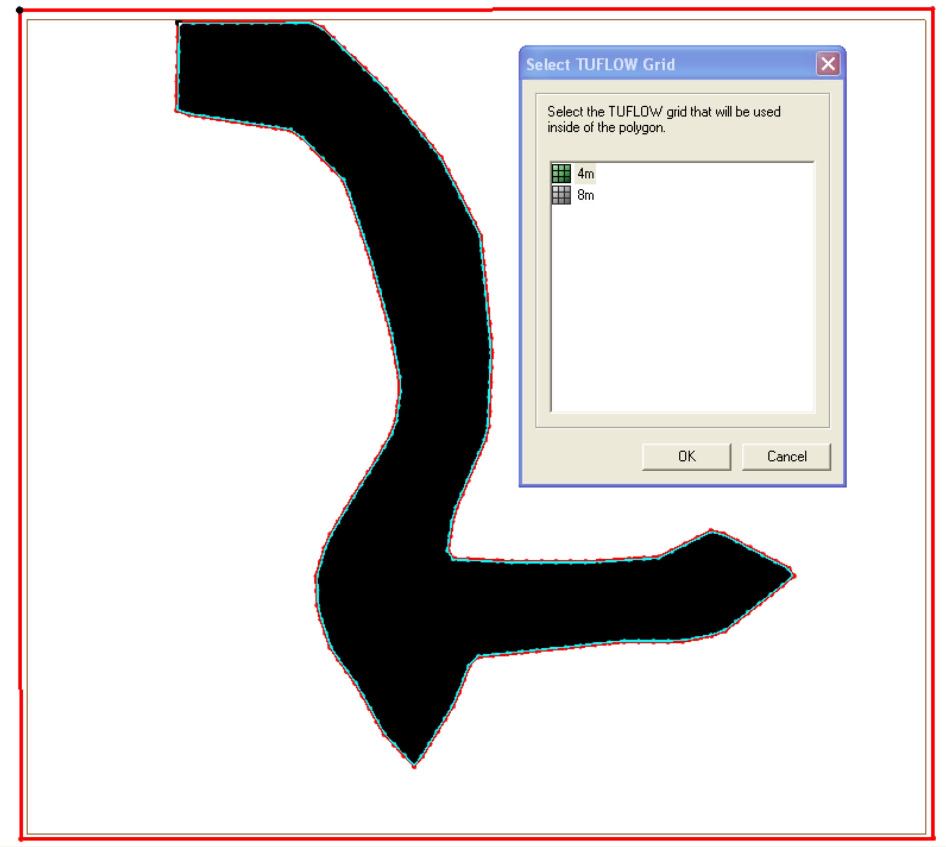
Any number of grids of varying sizes and/or orientations may be used in TUFLOW. Using multiple 2D domains requires that multiple 2D domains are licensed in TUFLOW.

To setup multiple 2D domains in SMS:

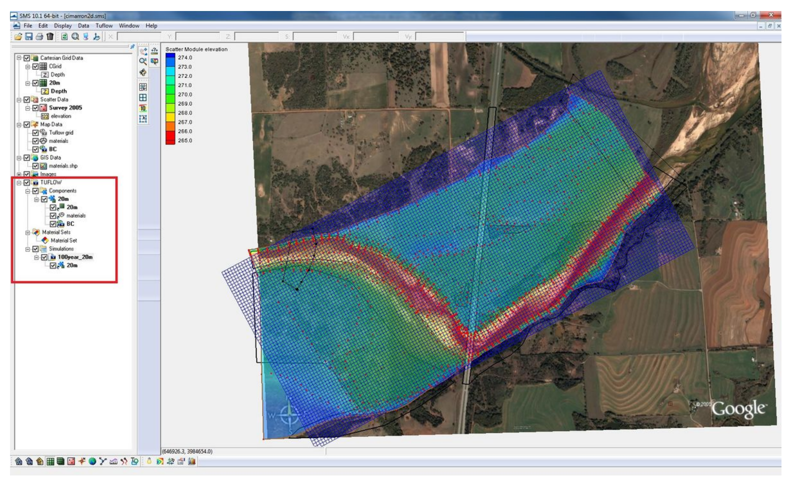
1. Create multiple TUFLOW grids and associated grid components.
2. Define a 2D/2D Linkage coverage and attributes (see below).
3. Add all the 2D domains being used and the 2D/2D Links Coverage to the simulation.

There are a few important items to keep in mind when setting up a 2D/2D Linkage coverage.

1. You need a polygon enclosing each domain you wish to use. In the case below, you would need a polygon that covers the area outside of the main channel polygon.
2. Each polygon needs to be linked to a specific 2D domain. This is done by double clicking on each polygon and specifying the appropriate 2D grid.
3. Set the vertex spacing on the arcs that join multiple 2D domains. The spacing along these arcs determines how flows move between the domains. The rule of thumb to start with is to have the vertex spacing equal to 1.5 times the larger cell size.



TUFLOW Links


TUFLOW Links

A TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. SMS allows for the creation of multiple simulations and each includes “links” to these items. Links are like shortcuts in windows and they call for the geometry component as well as each coverage used that is not part of the geometry component.

By using links the data is not duplicated; it just knows where to go to get the data that has to be used for each simulation. The use of links allows these items to be shared between multiple simulations. Multiple simulations can be created if some components need to be changed to create different scenarios and links would call for the same components used in previous simulations. Simulations also store the model parameters used by TUFLOW.

Links are located in the Project Explorer under TUFLOW Simulation once a simulation has been created.

TUFLOW Manholes

Automatic Manholes

Global manhole attributes can be set up for a TUFLOW simulation in the TUFLOW 1D model control. TUFLOW will then automatically create manholes when the following criteria are met:

- There is at least one incoming and one outgoing culvert – a culvert is a C, I or R channel.
- There are no open channels connected (ie. no bridges, weirs or any other channel that is not a culvert).
- Pit channels can be connected, but are not included in any of the calculations for determining manhole energy losses.

The following global attributes are set:

- **Loss Approach** – loss approach to be used (none, Engelhund, Fixed)
- **C Exit Coefficient** – K coefficient for flow out of circular manholes
- **R Coefficient** – K coefficient for flow out of rectangular manholes
- **Side Clearance** – for circular manholes, the default side clearance from the side of the largest culvert to the side of the manhole chamber. For rectangular manholes, the side clearance from the side of the culvert to the side of the manhole chamber
- **Type**– circular, rectangular, junction, or automatically determined
- **K Maximum Bend/Drop** – maximum K energy loss coefficient that can occur for the sum of the loss coefficients for bends and drops at a manhole when using the Engelhund approach
- **Minimum Dimension** – minimum diameter for circular manholes and minimum width and length for rectangular manholes
- **Manholes at All Culvert Junctions** – whether or not to have TUFLOW automatically create manholes.

Manual Manholes

The global manhole parameters can be overridden by creating manholes in a TUFLOW 1D Network coverage. Toggling on “Manhole” and selecting the “Attributes...” button in the 1D Network node properties dialog allows the user to set the following parameters for manually created manholes:

- **ID** – set a name for the manhole
- **Type** – circular, junction, rectangular
- **Loss Approach** – loss approach to be used (none, Engelhund, Fixed)
- **Invert** – bottom or bed elevation of manhole
- **Flow Width** – The flow width in meters (this is the diameter for circular manholes)
- **Flow Length** – The flow length in meters (not used for circular manholes)
- **Additional Storage Surface Area** – usually set to zero
- **K Fixed** – Set the fixed component of the calculated manhole loss coefficient
- **Km** – exit coefficient used by Engelhund approach
- **K Bend Max** – Upper limit of the combination of K theta and K drop used by Engelhund approach

If a TUFLOW 1D Network coverage has manholes defined, the manholes will be written out as their own layer when exporting TUFLOW files.

(TUFLOW Manual 2010)

TUFLOW Material properties

Material properties include roughness and hydrologic losses (if used). TUFLOW uses a manning n value for roughness which can be specified as a single parameter or it can vary with depth. Hydrologic losses include an initial loss (in mm) and a continuing loss rate (in mm/hr). Hydrologic losses are generally only necessary when using direct rainfall on the model domain.

Material Sets

Multiple definitions of material properties can exist in a project. Each definition includes the material properties for each material in SMS and is referred to as a "material set." A material set can represent roughness for different conditions such as winter/summer conditions or can be used in a sensitivity analysis. To build a material set, the TUFLOW solution must already be built. Right click on the tree item labeled "Material Sets" and choose "New Material Set."

Editing Material Properties

Material properties can be edited by right-clicking on the material set in the project explorer and choosing "Properties."

TUFLOW Menu

The following menu commands are available in the TUFLOW Menu:

Command	Functionality
Cell Attributes	This command is enabled when one or more cells on a TUFLOW grid are selected. The attributes that can be set using this command are the cell code and the option to override the global initial water level (set in the 2D Model Control). The cell code (water cell, inactive cell - in mesh, inactive cell - not in mesh) for the selected cells will only be used if “Specify cell by cell” is selected in “Cell codes” section of the TUFLOW Grid Options dialog (link).
Assign Material Type	This command is enabled when one or more cells on a TUFLOW grid are selected. The Materials Data dialog will allow the user to select a material to assign to a cell. The material set for the cell will only be used if “Specify cell by cell” is selected in the “Materials” section of the TUFLOW Grid Options dialog (link).

The following menu commands are available in the TUFLOW Simulation Menu:

Command	Functionality
Delete	This command will delete the TUFLOW simulation.
Duplicate	This command will create another copy of the simulation.
Rename	This command will allow you to rename the simulation.
Export TUFLOW files	SMS will save the TUFLOW input files in the TUFLOW directory for that simulation. If one does not already exist, the user will be prompted to select a location.
Launch TUFLOW	SMS will launch TUFLOW using input files that have previously been exported.
Save Project, Export and Launch TUFLOW	SMS will save the project, write out the TUFLOW input files, and launch TUFLOW.
2D Model Control	This command will bring up the 2D Model Control dialog where various model parameters are set. See TUFLOW 2D Model Control page (link).

Model Check	This command will cause SMS to perform a check for invalid settings that could prevent TUFLOW from running properly. If errors are found, they will be reported in the Model Check dialog with information on how to correct them.

TUFLOW Model Parameters

Run control parameters are divided into two sections for TUFLOW simulations. The main model control dialog and the model control parameters that apply specifically to the 1D portions of the TUFLOW run. Right-click on the simulation and choose the appropriate menu command to edit either set of model parameters.

This document will try to highlight frequently used model control parameters. Please see the TUFLOW documentation for a description of those items left out or for more information on specific items.

Model Control

The main model control dialog sets parameters for the run in general and some 2D specific controls.

Time

This page is used to set the run times for the simulation. The optimum timestep size is determined by the courant number. The rule of thumb is that the timestep should be about half as many seconds as the grid cell size in meters. So a grid with 10 meter cells, should run with a 5 second timestep. The timestep may be reduced to increase stability, but generally problems indicate another type of problem.

Output Control

This output control tab allows the user to specify what two dimensional (spatially varied) data should be saved in the TUFLOW run. The tab is broken down into three sections. These include the "Map Output", "Output Datasets" and "Screen/Log Output" sections.

"Map Output"

This section allows the user to specify the format and frequency of output created by TUFLOW. The user specifies:

- Format – Choose between XMDF and binary data (*.dat) file format or both. (Map Output Format line in the *.tcf file)
- Format Type – Choose between normal grid resolution (SMS 2dm), double normal grid resolution (SMS high resolution), or reaveraged grid resolution (SMS low resolution fancy). (Map Output Format line in the *.tcf file)
- Start Time – Choose the simulation time in hours at which spatially variable dataset output starts. (Start Map Output line in the *.tcf file).
- Interval: Choose the time interval in seconds between times steps in spatial datasets saved by TUFLOW (Map Output Interval line in the *.tcf file).
- Minimums/Maximums – TUFLOW can compute minimum and maximum values or only the maximum values for the simulation. This can be very handy in visualizing flood extents. (Store Maximums and Minimums line in the *.tcf file).
- Mass Balance Output toggle – turn this toggle on to have TUFLOW save the mass error (%) or cumulative mass error (%). The specific option must also be turned on in the "Output Datasets" section. (Mass Balance Output line in the *.tcf file).
- Zero negative depths – By default, TUFLOW zeros depths that are computed as negative values at cell locations. If you turn this off, and setup the display options in SMS to only show positive depths you can see partially wet cells. (Zero Negative Depths in SMS line in the *.tcf file).

"Output Datasets"

This section controls which datasets should be created by TUFLOW. The list is long and getting longer with each revision of TUFLOW. Select the toggle next to the desired items. (Map Output Data Types line in the *.tcf file).

"Screen/Log Output"

By default TUFLOW outputs to the screen information for each time step. This information is also written to a log file (*.tlf). TUFLOW supports various options for this type of run time feedback that are controlled in this section of the output control tab.

- Show water level for a point toggle and location – This option allows the user to instruct TUFLOW to display the water level on the screen for the cell located at the specified location. (Display Water Level line in the *.tcf file).
- Display interval – this control allows the user to decrease the frequency of the output information during a TUFLOW run. A value of 0 or 1 outputs at each time step. A larger positive value decreases the frequency to one in the specified number of time steps. A value of -2 suppresses all output except negative depth warnings and a value of -3 suppresses all time step display (Screen/Log Display Interval line in the *.tcf file).
- The user or developer can now specify additional text in the TUFLOW model control output tab which will be treated as custom output types. This will be used by developers and users to specify output types that are not yet supported directly in the interface.

Wetting/Drying

See the FESWMS manual for information on these options. Generally the default options are fine.

Restart Files

Restart files can be used to start a simulation part way through a run. They are generally used to generate an initial condition for several simulations that start with the same boundary conditions but are later changed for different events. For example to evaluate 10, 20, and 100 year events in a tidal area it might be useful to run a couple tidal cycles of a normal event and save these to a restart file. The restart file could be used with the 10, 20, and 100 year events without rerunning the first couple tidal cycles.

Water Level

- Initial Water Level – All models need to have an initial water level. Any cells with a lower elevation than the initial water level will start out wet and the rest will start dry. A TUFLOW model does not require a fully wet domain when starting like FESWMS or RMA2. It is generally a good idea for the initial water level to match the downstream boundary condition at the start of a run. The global initial water level can be overridden on a local level using a 2D Spatial Attributes Coverage.
- Override default instability level – By default TUFLOW will assume that if the water rises 10 m above the highest elevation in the grid then the run is unstable. This value may be overridden to allow the water level to get higher than this without causing TUFLOW to abort.

Eddy Viscosity

Eddy viscosity is used to compute energy losses due to turbulence not modeled at the scale of the simulation. TUFLOW supports specification of a constant eddy viscosity or using Smagorinsky specification. When using the Smagorinsky specification, the coefficient is generally between 0.06 and 1.0. It is recommended to use the default eddy viscosity of 1.0 when using constant eddy viscosities.

BC

- BC Event Name – The boundary condition event to use is specified.
- SA Minimum Depth – This value sets the minimum depth a wet cell must have to apply an SA inflow.
- SA Proportion to Depth – This allows the user to set whether or not to proportion SA inflows according to the depth of water.

Materials

The material set to use is specified in this tab.

Misc

- Write Check Files – TUFLOW creates a number of check files that can be used to verify model inputs. TUFLOW will run faster if not writing check files.
- Check inside grid error setting – By default TUFLOW will generate an error of entities exist outside the grid domain. If you want to use data that extends beyond your grid, you can change this to warning or off.
- Read External File – The SMS interface does not support every option available in TUFLOW. Unsupported options can be used by creating a TUFLOW command file (see the TUFLOW documentation) and specifying the file name here.
- Write Z pts as Binary Files (*.xf)
- TUFLOW Executable – This option determines which executable to use when running the simulation. The options are double and single precision for both 32 and 64bit. By default, it is the option chosen in the startup preferences.
- Use Mass Balance Corrector – This option sets whether or not TUFLOW should use the mass balance corrector, which carries out an additional iteration of the mass balance equation every half time step. This can result in significant reductions in mass error for problematic models, particularly those with steep and/or very shallow flow.

1D Control

General

- Timestep – The optimum 1D timestep is based upon the courant number. The 1D timestep can generally be made quite small without affecting run times.
- Output Interval – How often to write 1D solution data to the csv solution files.
- Initial Water Level – The initial water level should generally be the same as the 2D water level.
- Write Check Files – If this is off TUFLOW will not write check files for the 1D data.
- Read external estry control file – The SMS interface does not support all of the options available for 1D controls. You can build an estry control file (see the TUFLOW documentation for instructions) and have it read at the end of the SMS generated files.

Network

- Depth Limit Factor – This determines the depth that TUFLOW will consider unstable and abort the run. The default value is 10, meaning that if the channel depth is 10 times larger than the depth of the channel the simulation will be considered unstable.
- Conveyance calculation – For 1D cross-section based channels, the conveyance calculations are divided into subsections. The subsections can be divided anywhere there is a change in resistance or a channel can be formed around each channel point(All parallel option). If the all parallel option is not used it is possible for the conveyance in a section to decrease when the depth increases which gives TUFLOW and error.
- Minimum nodal area – Nodes with a small storage area can have stability problems. Adding a minimum nodal area for all nodes has the potential to make the run more stable but may attenuate the model solution.

TUFLOW Network Node SX Additions

These updates are intended for the 1D Network Coverage section of the TUFLOW Coverages page.

The TUFLOW documentation sections *4.5.1.3: Connecting Pits and Nodes to 2D Domains* and *4.5.3: 1d_nwk Attributes* list some new TUFLOW features that SMS now supports. SMS's Network Node Attributes dialog has several new additions to the Create Connection to 2D Domain (SX) section. These new options allow you to control elevations at the connections, how many cells are connected, and the method for selection of additional cells (Grade or Sag). Each option correlates fairly directly to a TUFLOW field and some are labeled as such to make lookup easy.

TUFLOW Numeric Engine

TUFLOW is a computational engine that provides two-dimensional (2D) and one-dimensional (1D) solutions of the free-surface flow equations to simulate flood and tidal wave propagation. It is specifically beneficial where the hydrodynamic behaviour in coastal waters, estuaries, rivers, floodplains and urban drainage environments have complex 2D flow patterns that would be awkward to represent using traditional 1D network models.

A powerful feature of TUFLOW is its 2D/1D dynamic linking, first pioneered in1990, and subsequently enhanced to the point where it offers unparalleled flexibility and robustness.

TUFLOW continues to develop and evolve to meet the challenges of hydrodynamic modelling. Its strengths include:

- rapid and stable wetting and drying;
- 1D and 2D linking;
- multiple 2D domains (optional);
- both 1D and 2D representation of hydraulic structures;
- automatic upstream/downstream controlled flow regime switching;
- 1D and 2D supercritical flow;
- highly flexible and efficient data handling;
- GIS based; and
- extensive quality control outputs.

It is suited to modelling flooding in major rivers through to complex overland and piped urban flows, along with estuarine and coastal hydraulics. [3]

TUFLOW was written and is maintained by Bill Syme at WBM TUFLOW Webpage [2].

TUFLOW Running

Before running TUFLOW, the TUFLOW files must be generated from SMS. These files are separate from the files SMS uses to store its TUFLOW information.

The options to export the TUFLOW files and run TUFLOW are in the right-click menu of the simulation.

- Export TUFLOW files – This will export the TUFLOW model files into the run directory (by default a directory named TUFLOW under the project directory).
- Launch TUFLOW – This will launch TUFLOW under the assumption that the files have already been exported as above.
- Save Project, Export TUFLOW files, and Launch TUFLOW – This option will save the current SMS project file, export the TUFLOW files, and launch TUFLOW on these files.

TUFLOW Simulation

A simulation includes the domain, boundary conditions, model parameters, event definition, and material set used for a single TUFLOW run. Multiple simulations can exist within the same project in order to compare alternatives such as before/after a levee or to look at multiple events such as 10 year, 20 year, and 50 year.

Working with Simulations

A Simulation can be created by right-clicking on a blank portion of the project explorer and choosing "New → TUFLOW Simulation." Once a simulation exists future simulations are often similar to one already created. Often it is easier to make a copy of the simulation and make changes where appropriate. A simulation can be copied by right-clicking on the simulation and choosing "Duplicate."

The right-click menu for a simulation includes commands to change model parameters or 1D model parameters as well as launching TUFLOW to run the simulation.

Simulation Components

Much of the data for a simulation is contained in Geometry components and TUFLOW coverages. These items are included in a TUFLOW simulation using project explorer links. The model parameters are stored with the simulation item and can be accessed from the simulation's right-click menu.

TUFLOW Viewing Results

TUFLOW results are written in SMS mesh format in the \TUFLOW\results folder. The mesh module tools and menus are used to visualize the solution.

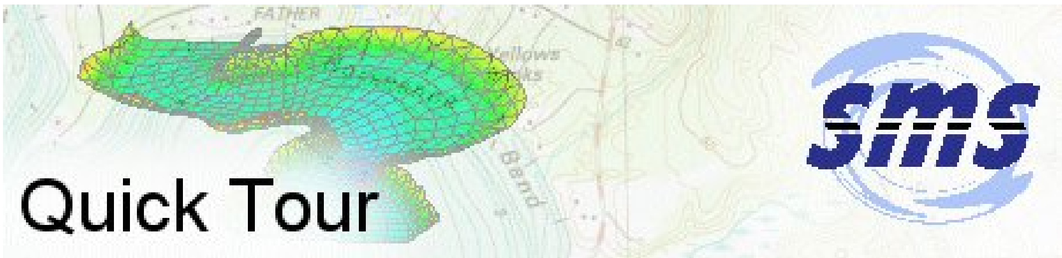
The solution files all start with the name of the simulation. The extension identifies the type of data in the result file.

- *.2dm – two-dimensional output mesh in SMS format.
- *.dat – The dataset files associated with the mesh solution (d for depth, h for water surface, etc.)
- *.mat – The output materials from TUFLOW. Most of the element materials come from the *.tmf file. TUFLOW also uses the materials to identify boundary cells.
- *.ALL.sup – A file that opens the *.2dm file, all the *.dat files, and the .mat file.
- *.hV.sup – A file that opens the *.2dm file, the water surface and velocity datasets, and the *.mat file.

7. Appendix

7.1. Quick Tour

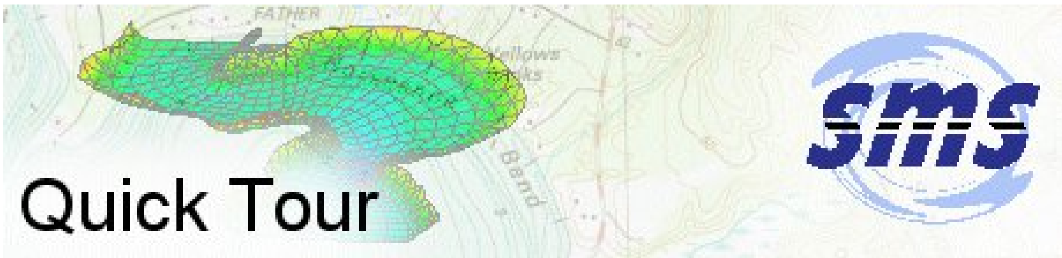
Quick Tour



The Quick Tour Section presents a brief introduction of SMS features:

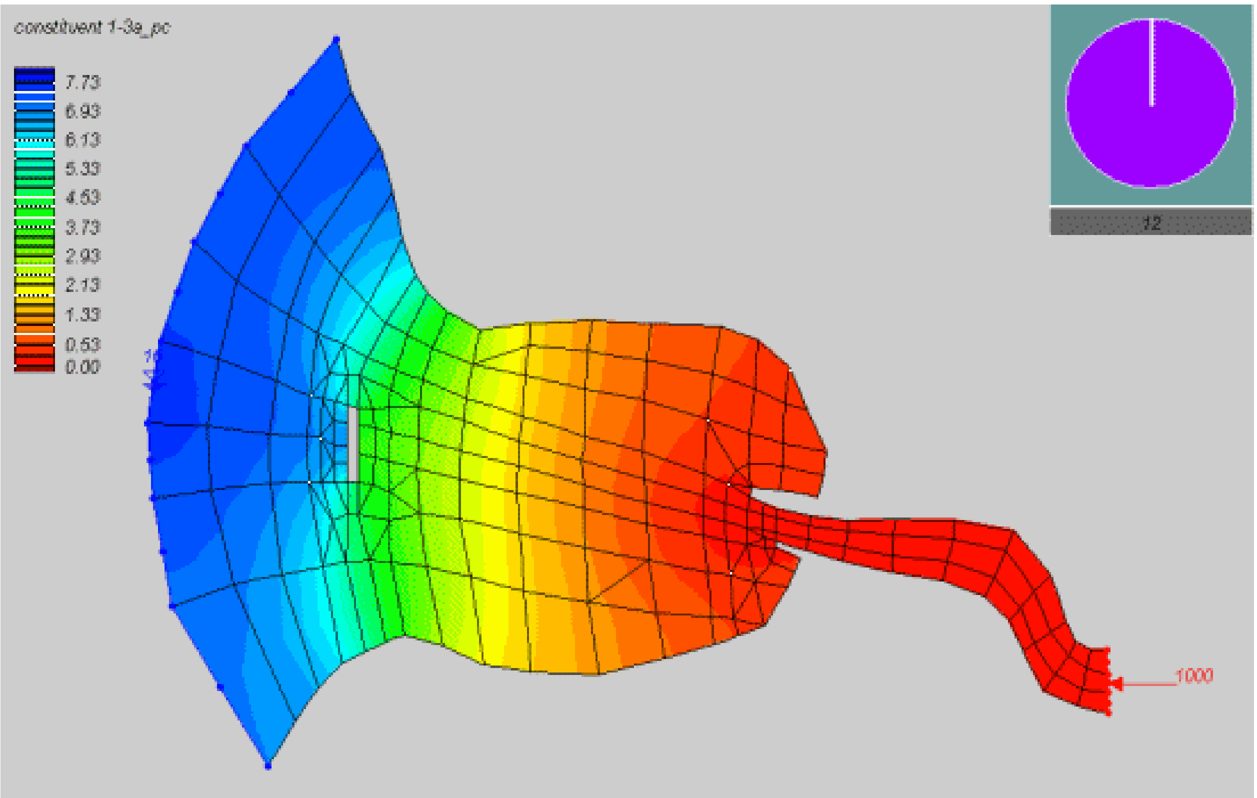
- Quick Tour – Modules
- Quick Tour – Animation
- Quick Tour – Background Images
- Quick Tour – Calibration
- Quick Tour – Calibration Tartgets
- Quick Tour – Conceptual Models
- Quick Tour – Coordinate Conversions
- Quick Tour – Datasets
- Quick Tour – Edit Strip
- Quick Tour – Feature Objects
- Quick Tour – Feature Stamping
- Quick Tour – FESWMS... to Learn More
- Quick Tour – FHWA
- Quick Tour – Filtering
- Quick Tour – Generic Model
- Quick Tour – Help Strip
- Quick Tour – Layout
- Quick Tour – Macros
- :Quick Tour – Menus
- Quick Tour – Model Conversion
- Quick Tour – Models... to Learn More
- Quick Tour – Network Creation
- Quick Tour – Observation Coverage... to Learn More
- Quick Tour – Polygon Attributes
- Quick Tour – Profiles
- Quick Tour – Project Explorer
- Quick Tour – Scatter Data
- Quick Tour – Support
- Quick Tour – Tools
- Quick Tour – USACE-WES Models
- Quick Tour – Visualization

Quick Tour Animation

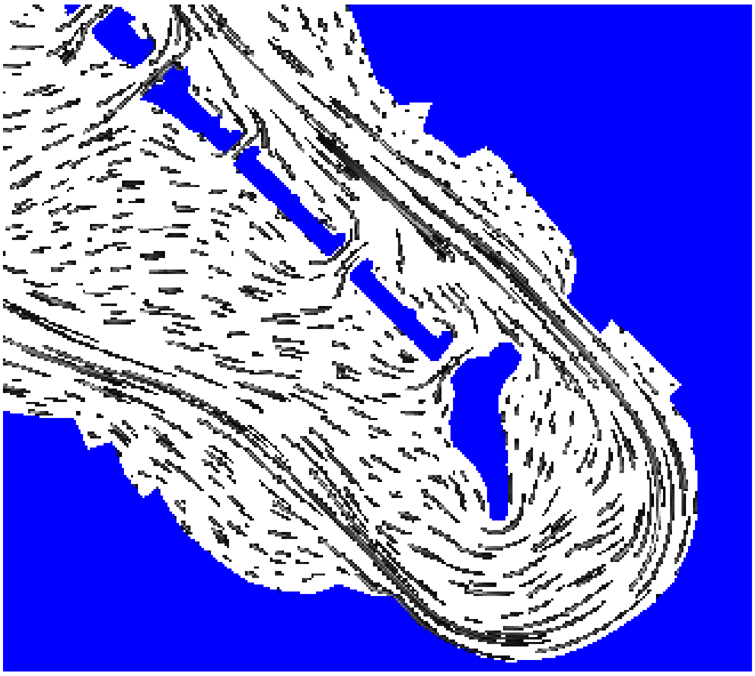


One of the most powerful visualization tools in SMS is animation. Animation film loops can be generated from any transient data set in seconds. The film loop can be played back in SMS or saved as a Windows video file (*.avi) file for inclusion in web pages (as shown below) or PowerPoint presentations.

Filmloop

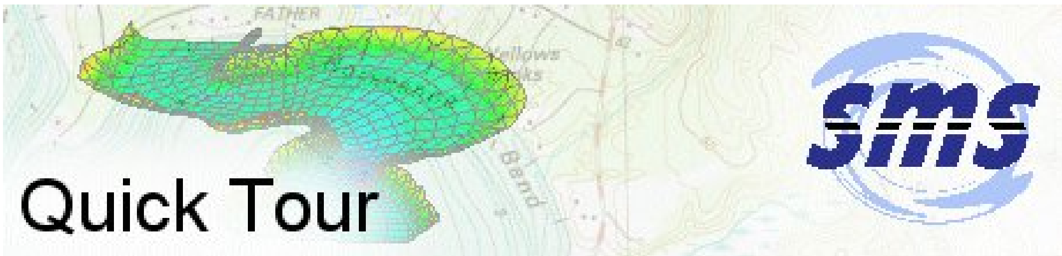


Flow Trace

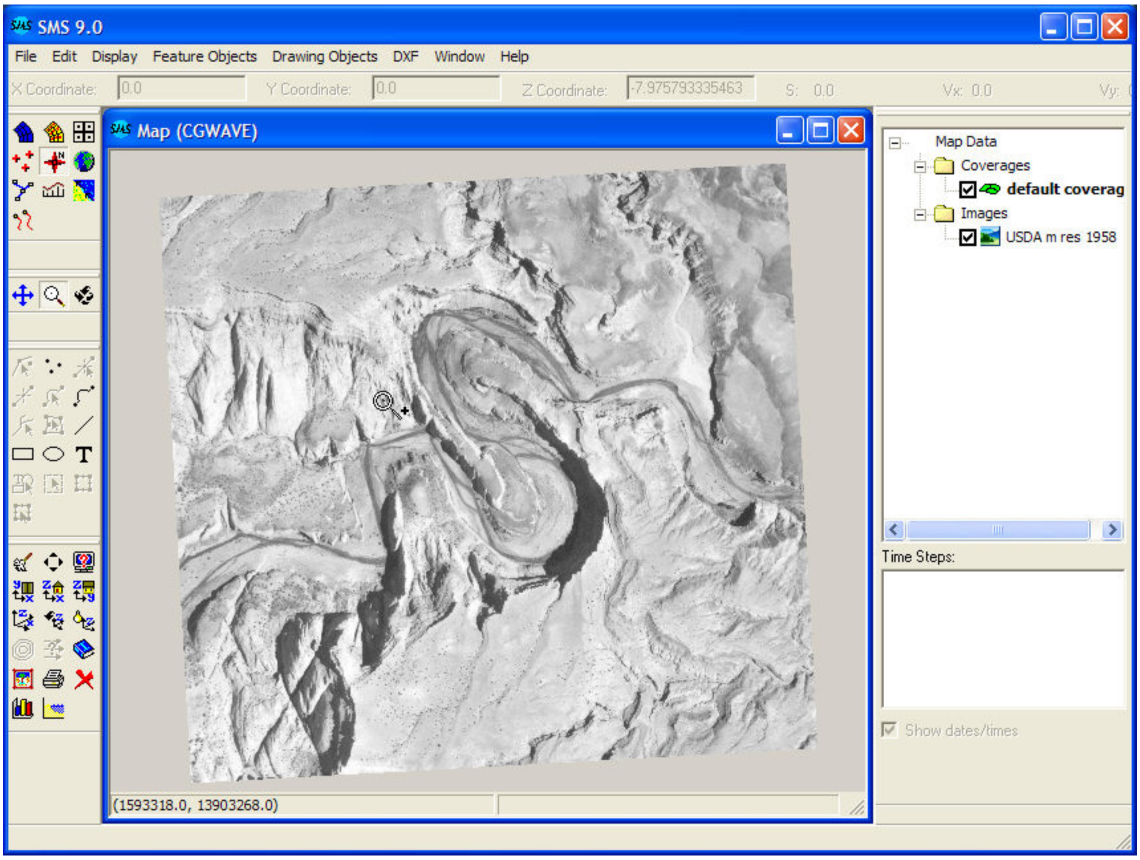


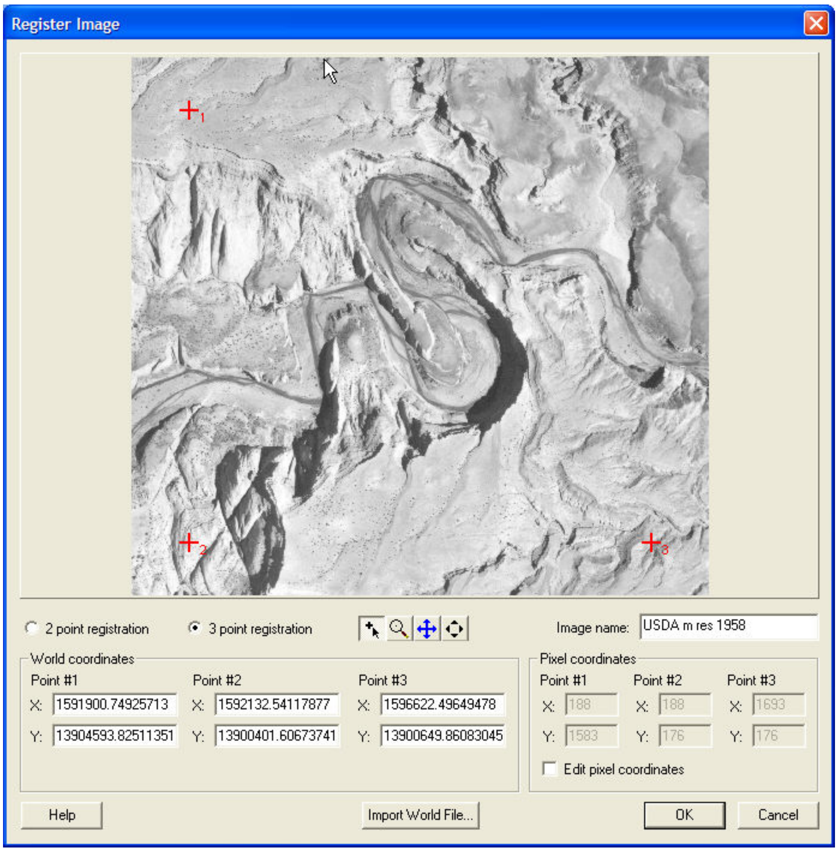
< Previous | Next >

Quick Tour Background Images



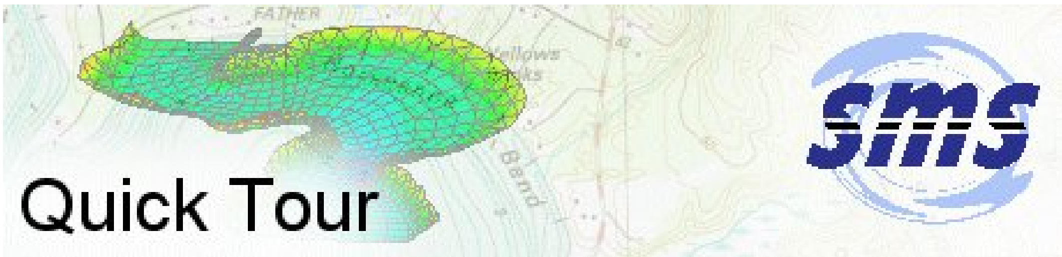
When constructing a model within SMS, the first step is often to import and register a scanned digital image or aerial photo. This makes it possible to perform on- screen digitizing and ensures that objects are created at the proper location.



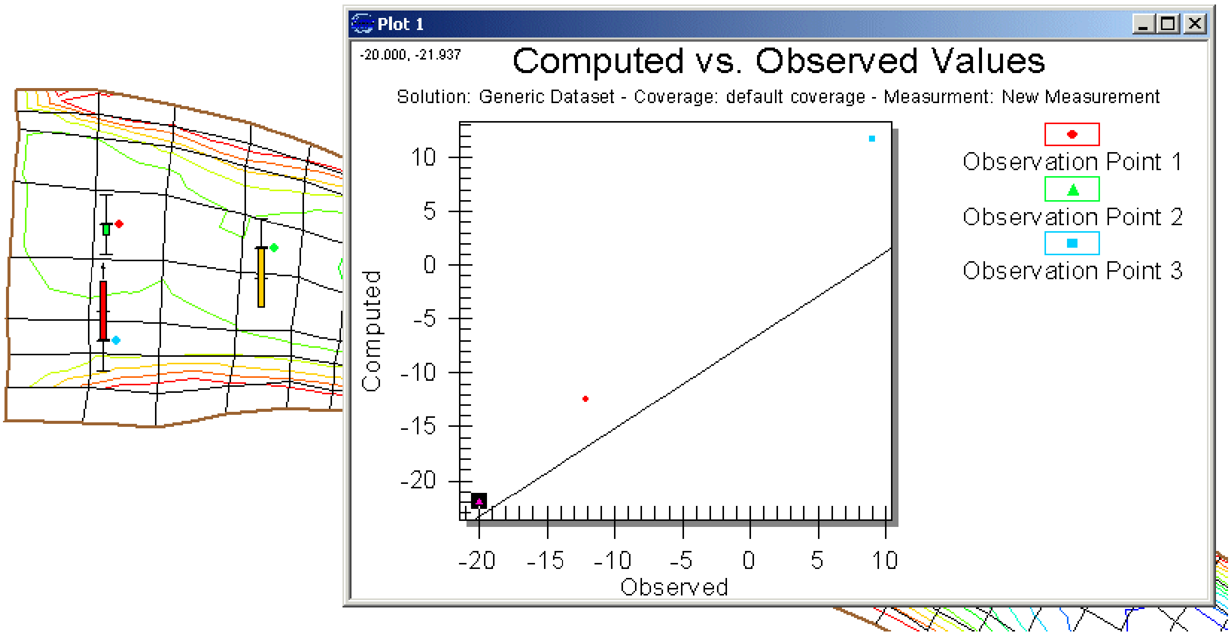


< Previous | Next >

Quick Tour Calibration

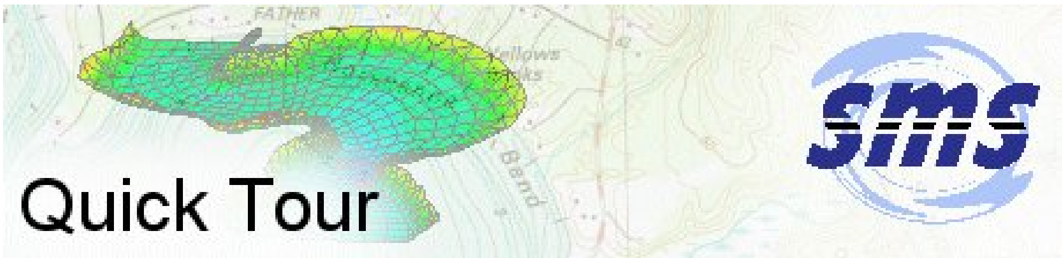


Model calibration involves changing input parameters until the model results match field observations. SMS provides a suite of options for model calibration including calibration targets and plots of calibration statistics.

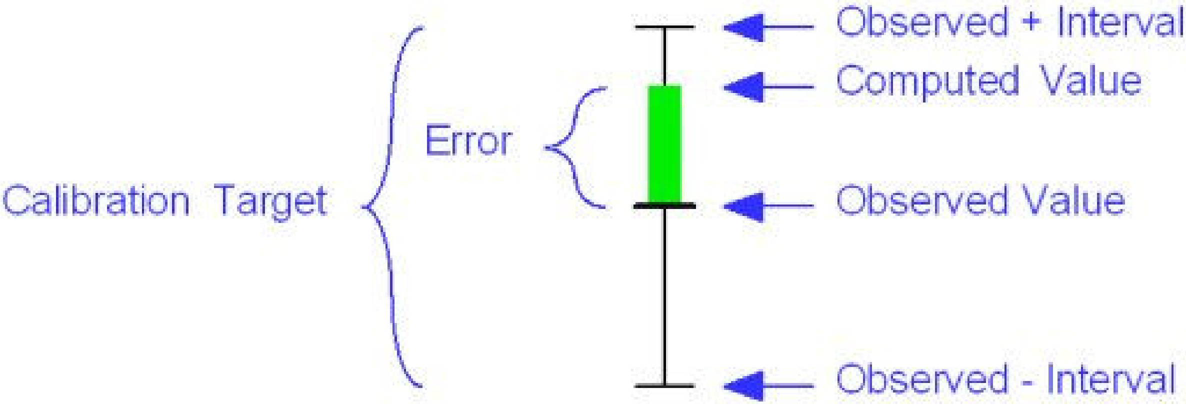


< Previous | Next >

Quick Tour Calibration Tartgets

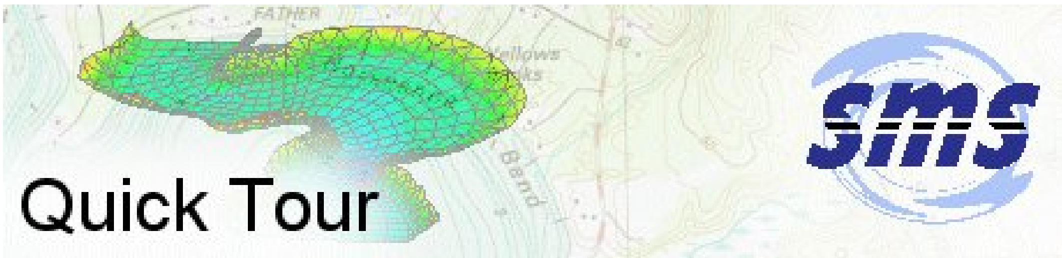


For both point and flux observations, SMS provides an option for plotting a calibration target at the location of the observation. The calibration target displays the relationship of the computed value to the measured value in a simple, intuitive manner.



< Previous | Next >

Quick Tour Conceptual Models



There are two main methods for building models in SMS, the direct approach and the conceptual modeling approach.

With the direct approach, the first step is to create nodes and then connect the nodes to form elements. The model parameters, source/sink data, and boundary conditions are assigned directly to the nodestrings, nodes, and elements of the mesh. Because the number of nodes and elements increases very quickly as the complexity of the project grows, this approach is only suited for very simple models.

The most efficient approach for building realistic, complex models is the conceptual model approach. With this approach, a conceptual model is created using GIS objects, including points, arcs, and polygons. The conceptual model is constructed independently of a mesh or grid. It is a high-level description of the site including geometric features such as channels and banks, the boundary of the domain to be modeled, flow rates and water surface elevations of boundary conditions, and material zones with material properties such as Manning's n value. Once the conceptual model is complete, a mesh or grid network is automatically constructed to fit the conceptual model, and the model data are converted from the conceptual model to the elements and nodes of the mesh network.

< Previous | Next >

Quick Tour Coordinate Conversions

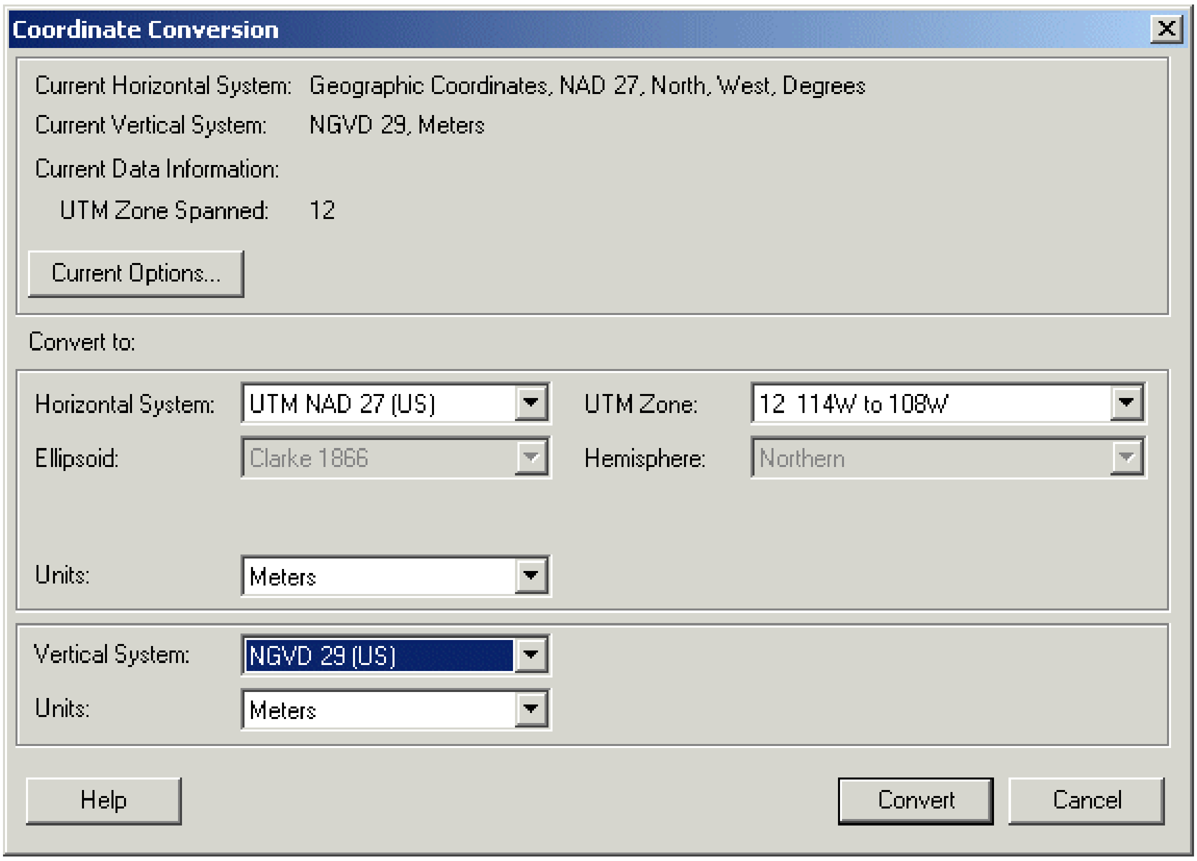
SMS has coordinate conversions that allow for converting any data contained in SMS to/from several coordinate systems, including:

Horizontal Coordinates

- Geographic
- UTM
- State Plane
- User Defined System

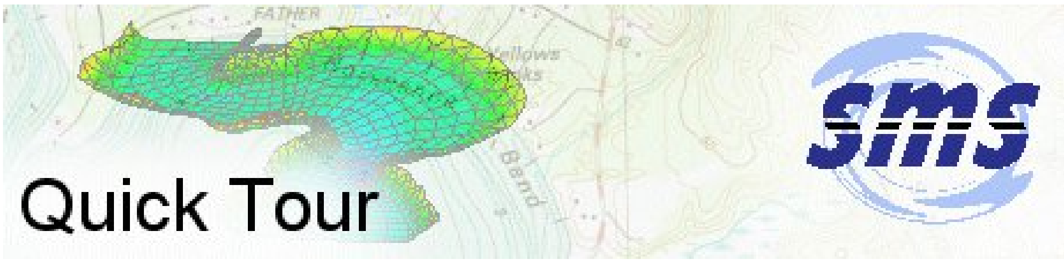
Vertical Coordinates

- NGVD 29
- NAVD 88

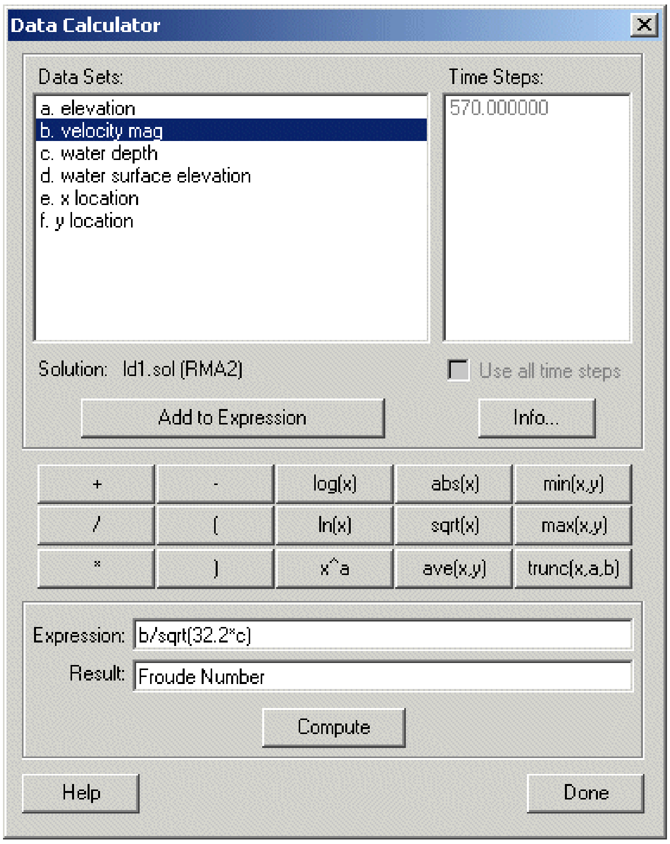


< Previous | Next >

Quick Tour Data Calculator

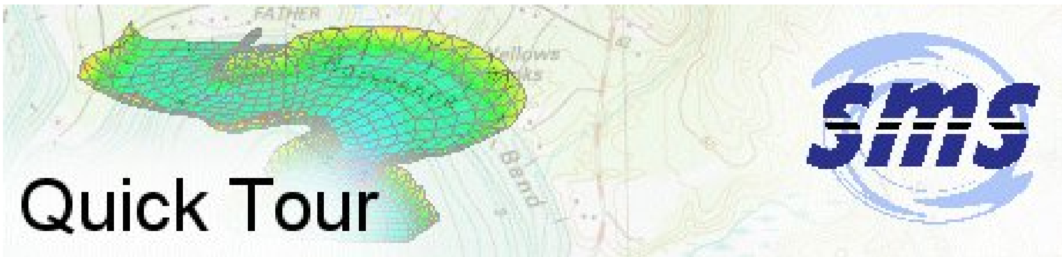


The Data Calculator can be used to enter mathematical expressions to generate new data sets from existing data sets. For example, the Data Calculator can be used to compute a data set representing the difference between two other data sets.

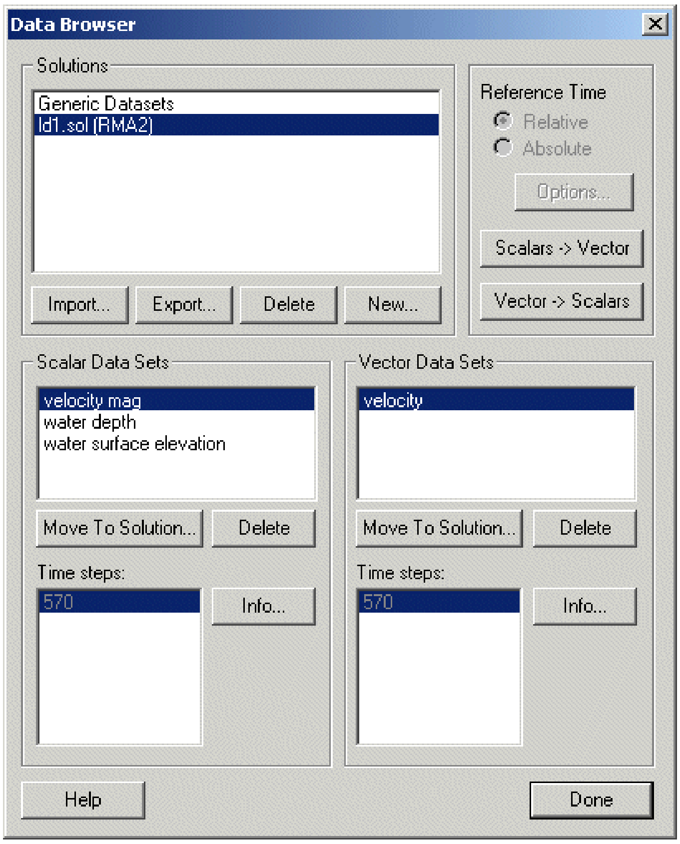


< Previous | Next >

Quick Tour Datasets

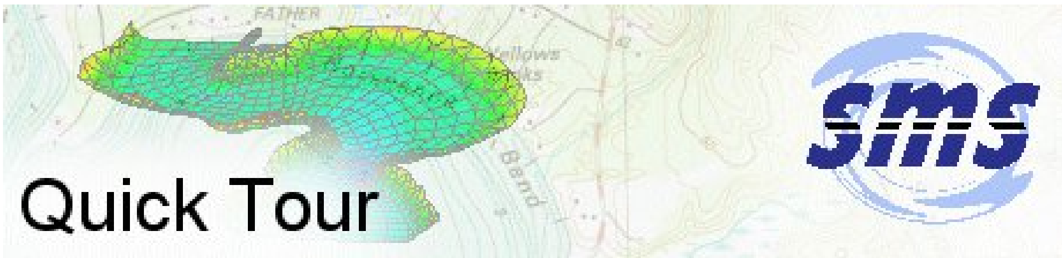


Large data arrays in SMS are organized as "data sets." A data set is a set of vector or scalar values associated with a mesh, grid, or scatter point set. Data sets can be steady state or transient (multiple time steps). Most data sets represent model solutions but data sets also result from interpolation and operations on existing data sets. Contour displays are based on the active data set for an object. Data sets are managed using the Data Browser and the combo boxes at the top of the SMS window.

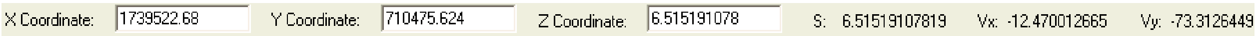


< Previous | Next >

Quick Tour Edit Strip

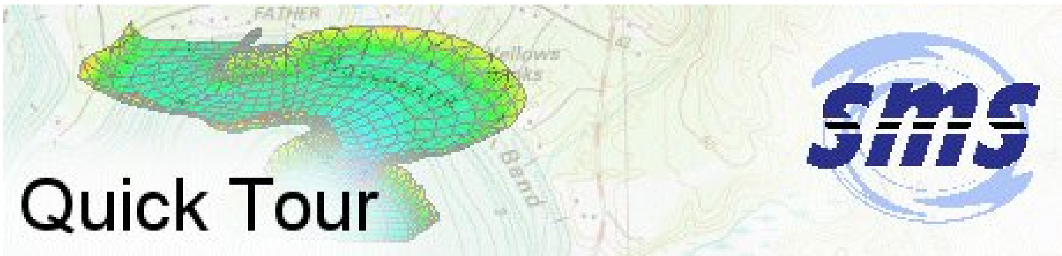


The edit strip below the menus allows the user to see the coordinates of the selected node, vertex, point, etc. In most cases, the user can edit the entity by changing the values of the x, y, and z coordinates. The data set values at that location are also displayed to the right.



< Previous | Next >

Quick Tour FESWMS... to Learn More

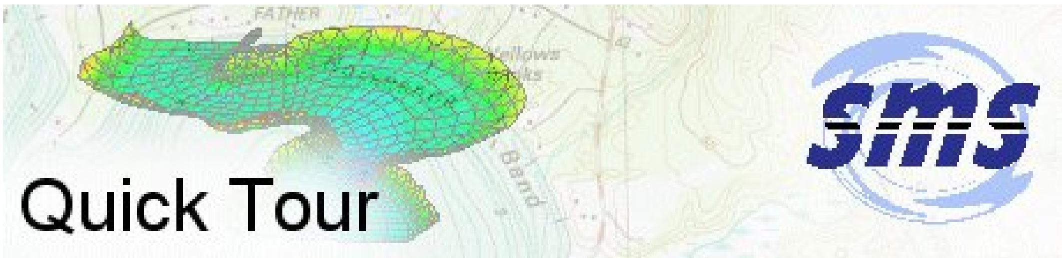


To learn how to set up model simulations in SMS, consult the following tutorial exercises in the SMS Tutorials document:

- Basic FESWMS Analysis
- Advanced FESWMS Analysis

< Previous | Next >

Quick Tour FHWA

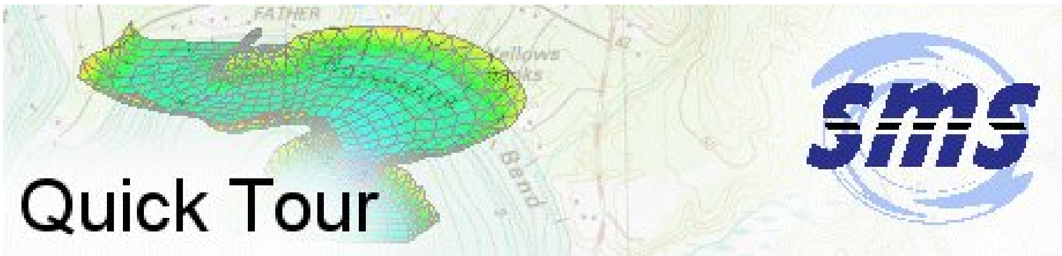


SMS includes interfaces to model supported and used by the United States Department of Transportation Federal Highway Administration. The FHWA models with interfaces in SMS include:

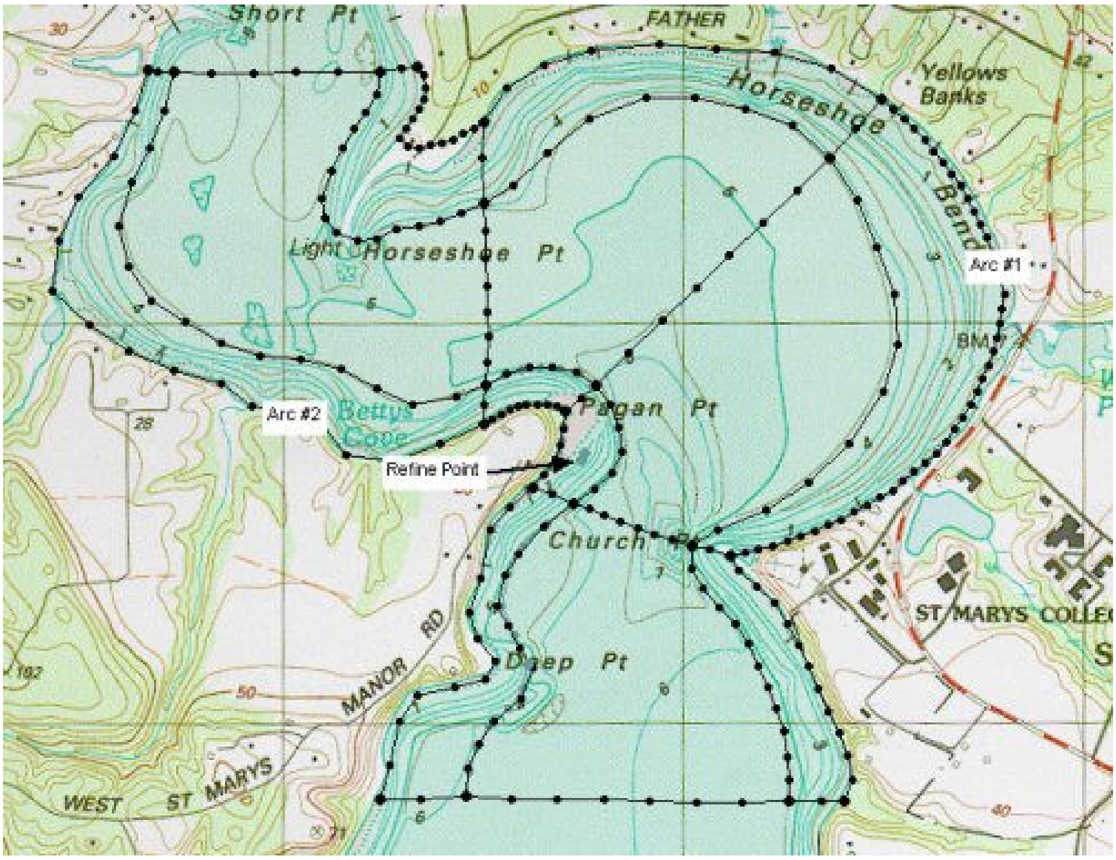
- **FESWMS** - FESWMS is a finite element hydrodynamic modeling code that supports both super and subcritical flow analysis, including area wetting and drying. The FESWMS model allows users to include weirs, culverts, drop inlets, and bridge piers in a standard 2D finite element model.

< Previous | Next >

Quick Tour Feature Objects

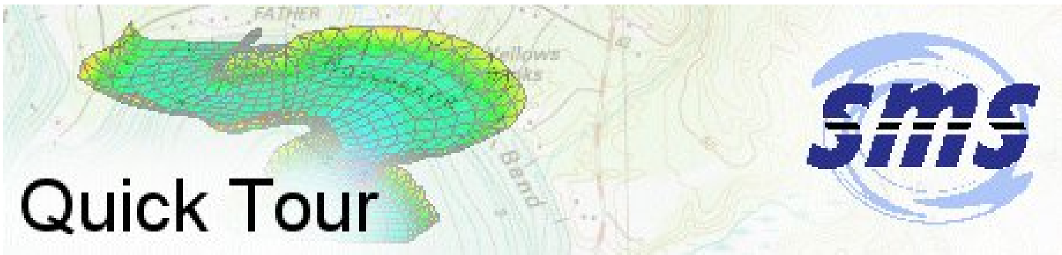


Conceptual models are constructed using feature objects. Feature objects are points, arcs, and polygons. Feature objects can be created entirely within SMS or imported from a GIS using shapefiles. Model parameters are assigned to the feature objects.



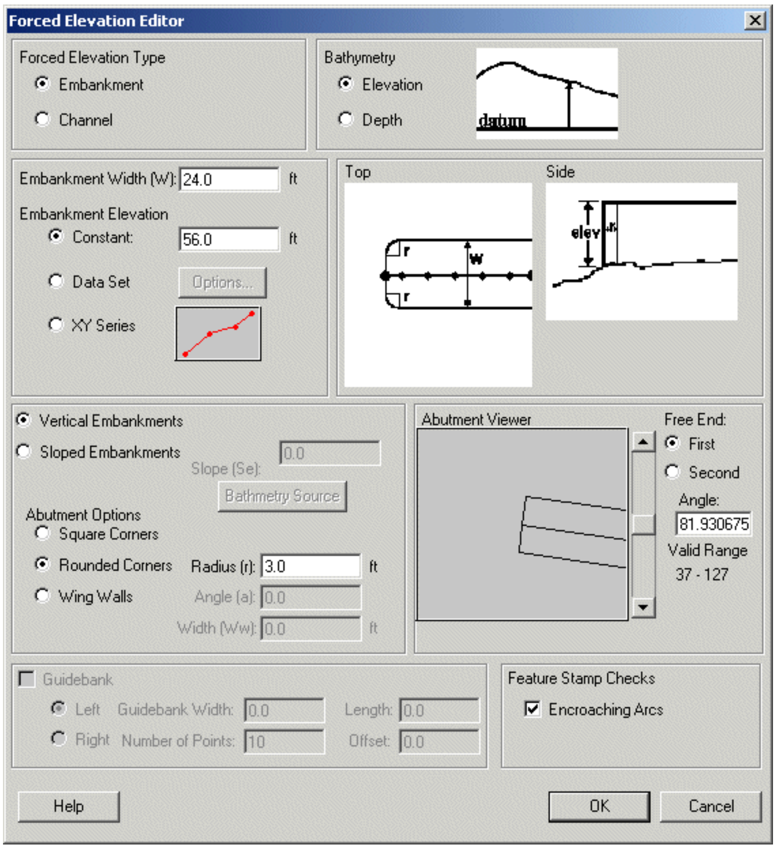
< Previous | Next >

Quick Tour Feature Stamping



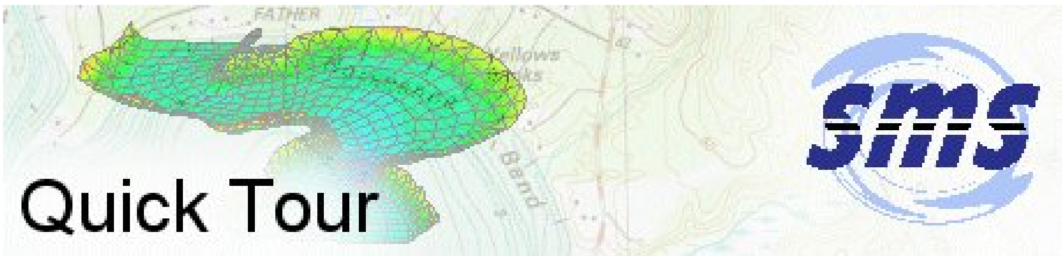
Some conceptual features may be desired in the numerical model that do not exist in the bathymetry of the model. Examples include jetties, breakwaters, bridge embankments, and dredged channels.

The feature stamping option in SMS creates the GIS entities to represent these features from a centerline arc or arc group and feature parameters.

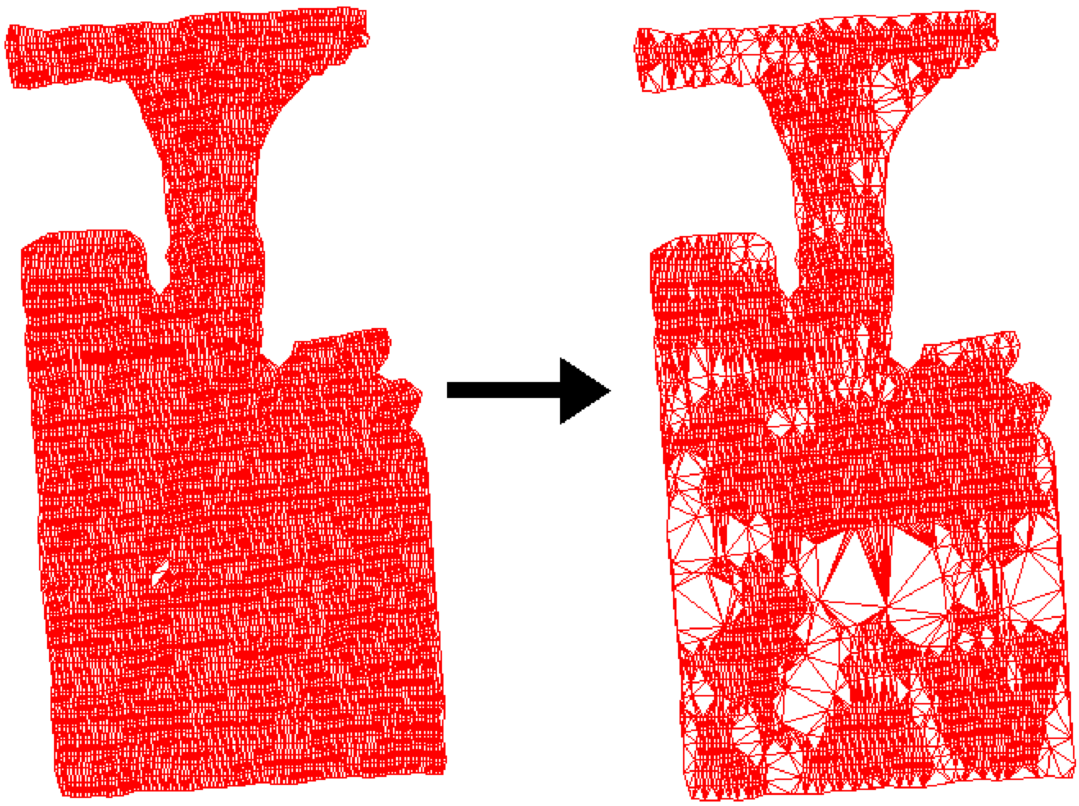


< Previous | Next >

Quick Tour Filtering

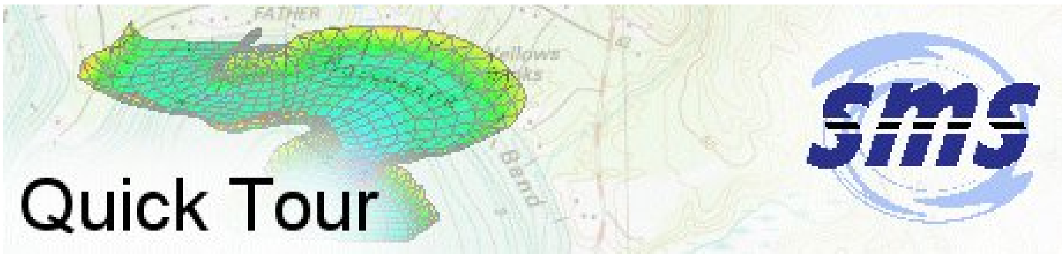


Filtering reduces dense scattered data by removing redundant points in planar regions.



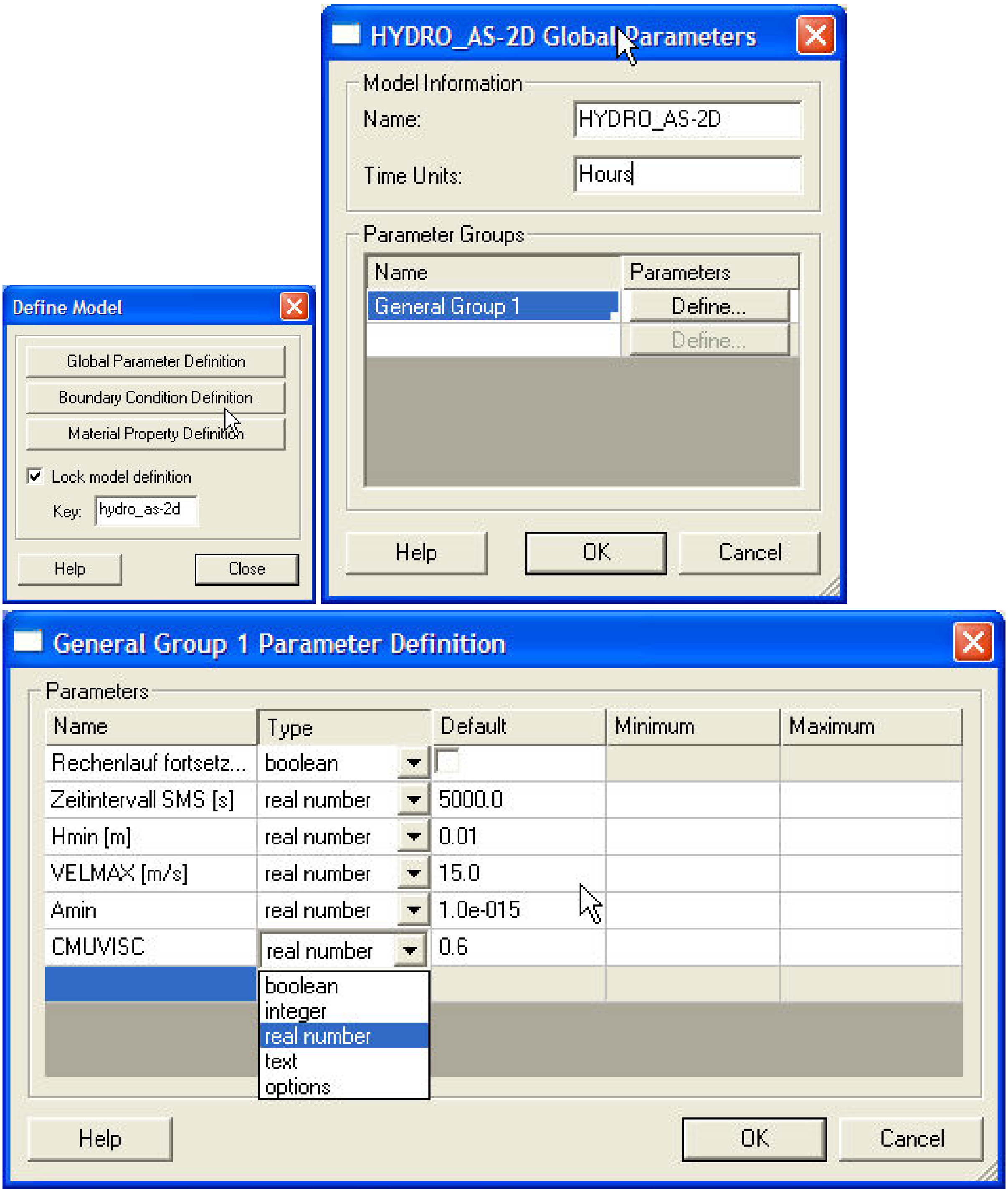
< Previous | Next >

Quick Tour Generic Model

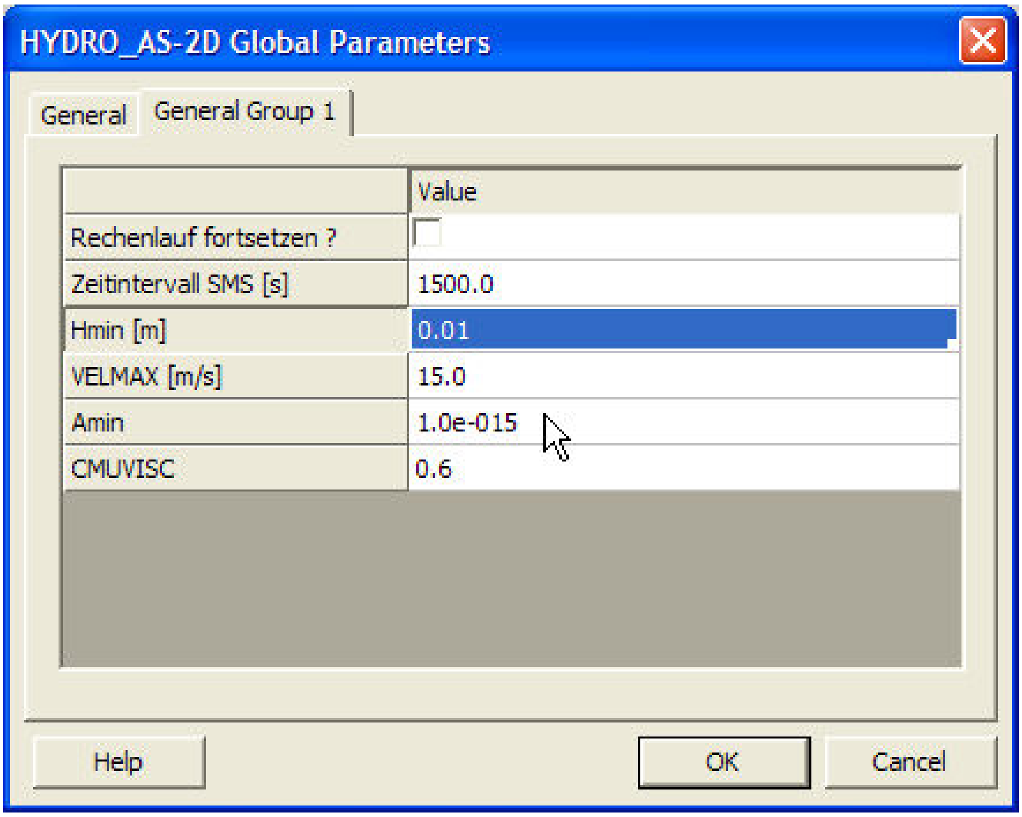


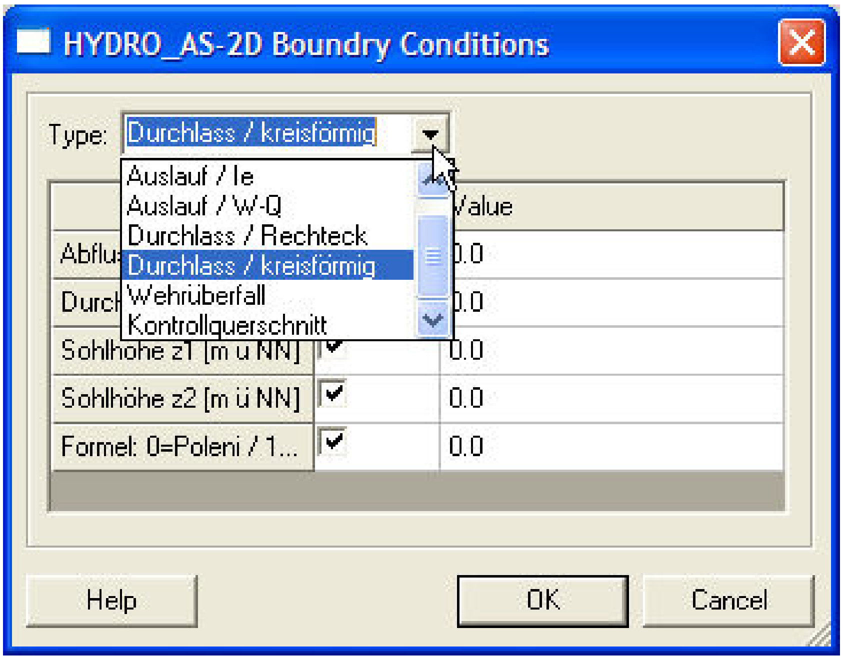
While SMS contains several custom interface for specific numerical models, it is impossible to build such interfaces for all possible models. Therefore, a generic interface has been incorporated into the system whereby any two-dimensional finite element or finite difference model can be run using SMS as the pre- and post-processor.

Model parameters are set up including model parameters, boundary conditions and spatial (material) properties. SMS saves these definitions for use in specific modeling applications.



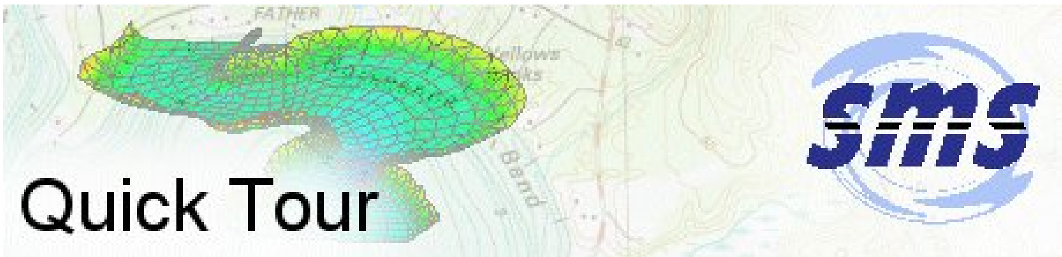
With the model attributes defined, the user can generate a mesh or grid using the geometric tools in the Map, Data, Mesh and Cartesian Grid modules. Then model parameters can be set, boundary conditions specified, and material properties applied to the geometric data and saved out to a generic file format.





< Previous | Next >

Quick Tour Help Strip

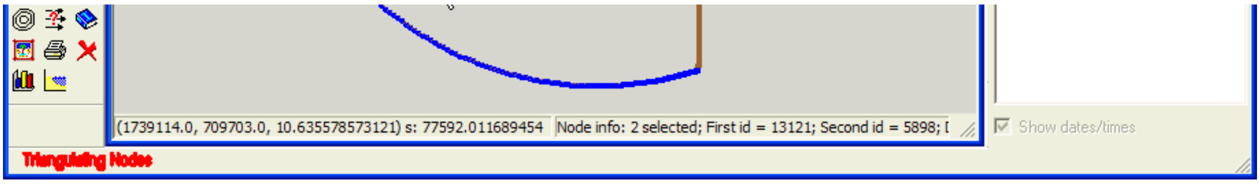


The help strip at the bottom of the SMS windows is used to provide feedback.

On the bottom left of the graphics window, the coordinates of the cursor and the function value of that point are displayed if appropriate. If a process in SMS will take a significant amount of time, a progress bar is also displayed here.

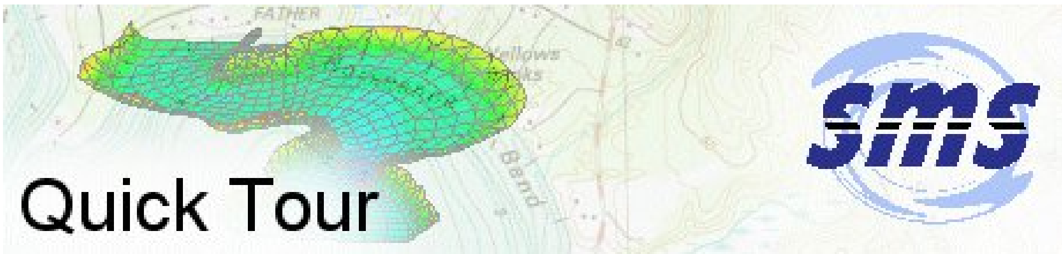
On the bottom right of the graphics window, the attributes of the selected entities are displayed including number of entities, distance between two entities, etc.

Along the bottom of the program window, SMS displays messages about what it is processing.

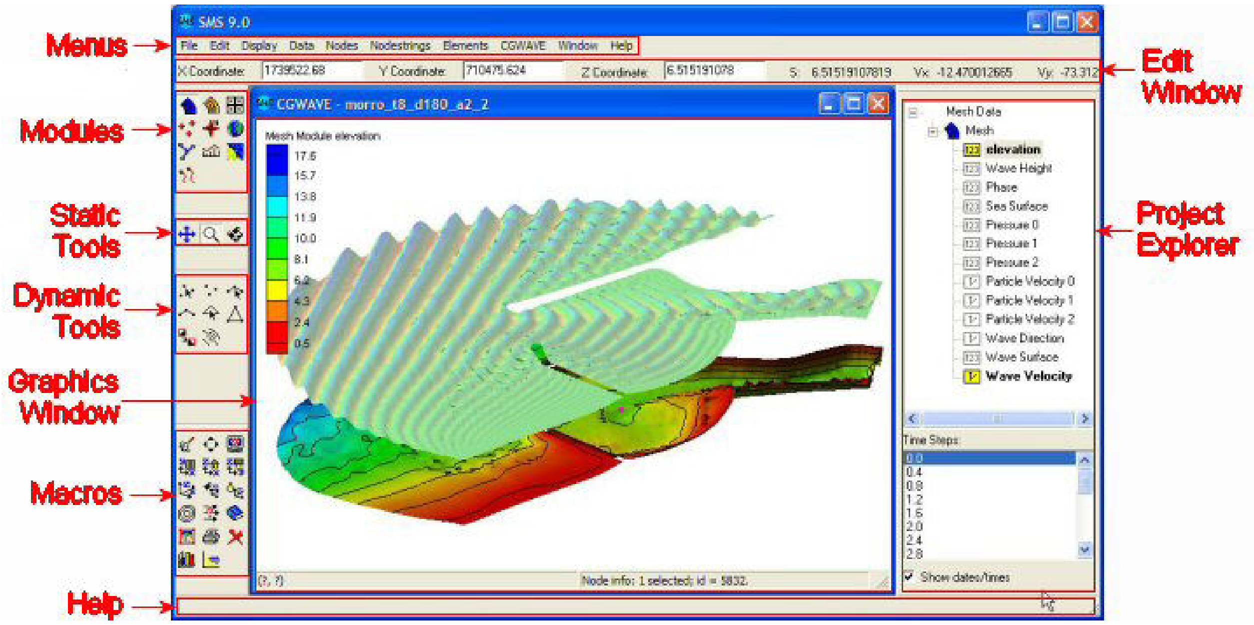


< Previous | Next >

Quick Tour Layout

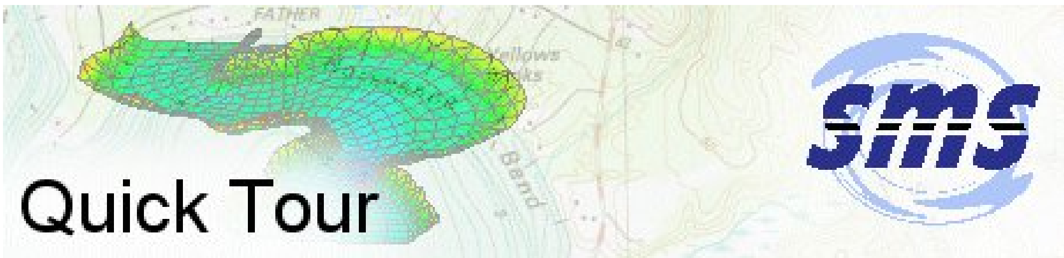


The SMS window is organized into the following sections:



< Previous | Next >

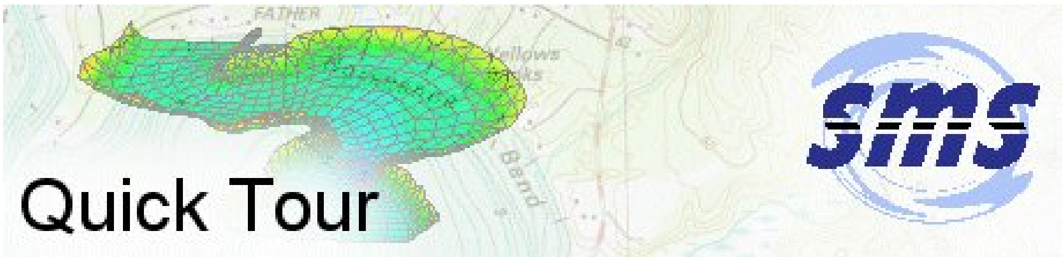
Quick Tour Macros

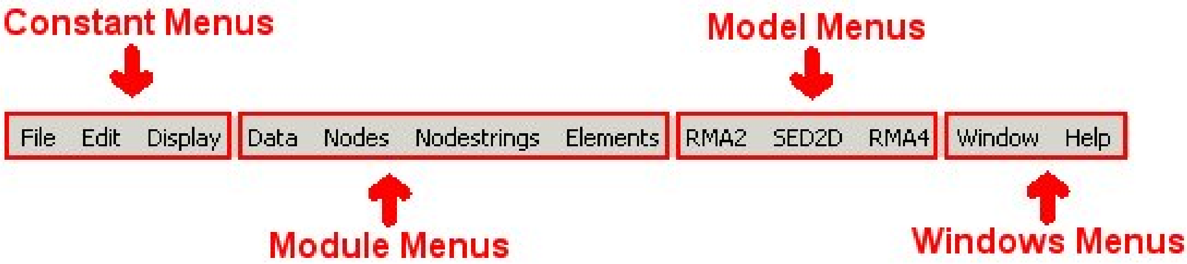


The macros are a set of icons at the bottom of the tool palette. Macros are shortcuts to frequently used menu commands.

< Previous | Next >

Quick Tour Menus



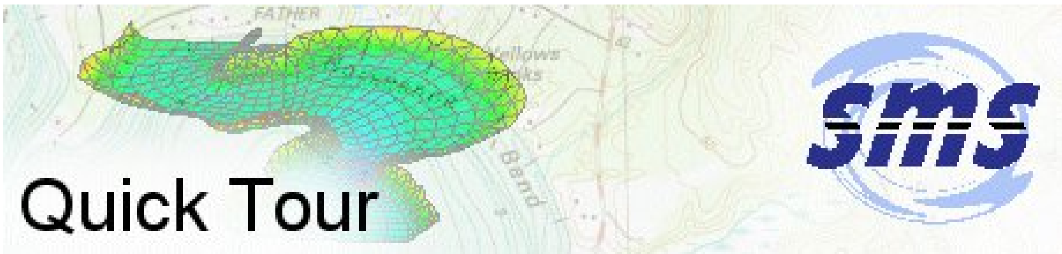


The first three menus are constant for all modules in SMS. The second group of menus allow operations on entities associated with the selected module. The third group of menus change based on the selected model. The last two menus are standard windows menus..

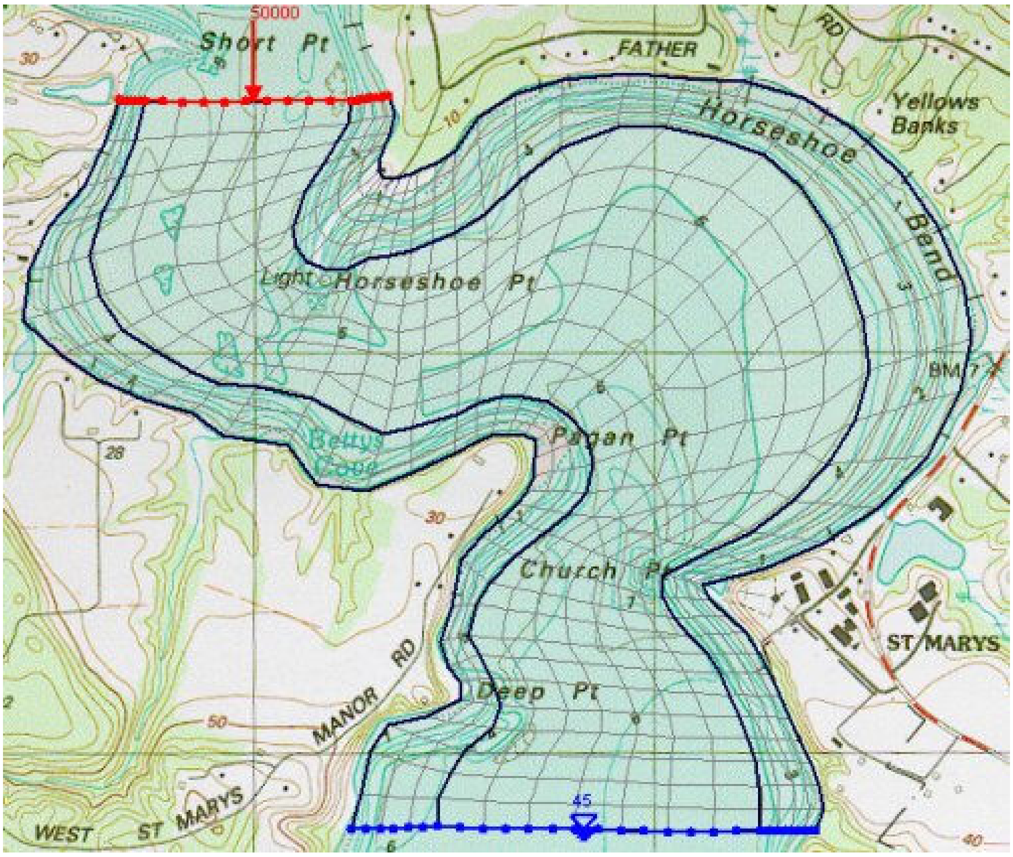
The set of menus shown in this section correspond to the mesh module with TABS as the current model.

< Previous | Next >

Quick Tour Model Conversion

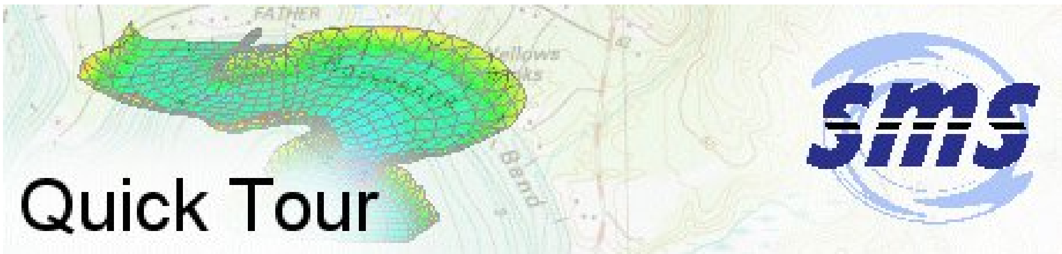


Once the network is created, the next step is to assign the attributes from the conceptual model to the network. This automatically assigns boundary conditions and material properties. If a change is made to the conceptual model, the network and its attributes can be updated in seconds.



< Previous | Next >

Quick Tour Models... to Learn More

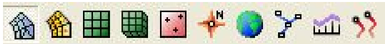


To learn how to set up model simulations in SMS, consult the following tutorial exercises in the SMS Tutorials document:

- BOUSS2D Analysis
- CGWAVE Analysis
- IMS-ADCIRC Analysis
- IMS-M2D Analysis
- Basic RMA2 Analysis
- Advanced RMA2 Analysis
- RMA4 Analysis
- STWAVE Analysis

< Previous | Next >

Quick Tour Modules



SMS is organized into modules. Each module is associated with a particular object type. Only one module is active at any given time. As you switch modules, the menus and the tools unique to the active module are displayed.

Mesh. Contains tools for editing unstructured networks. Includes tools for working with nodes, elements, nodestrings and model parameters.

Boundary Grid. Contains tools for editing boundary fitted structured grids. Includes tools for working with cells and cell vertices, cellstrings and model parameters.

Cartesian Grid. Contains tools for creating/editing Cartesian structured grids. Includes tools for working with cells and cell strings for cell centered grids, and nodes and nodestrings for mesh centered grids. Also includes tools for editing model parameters for Cartesian Finite Difference models.

3D Cartesian Grid. Contains tools for creating/editing Cartesian structured grids. Includes tools for working with cells and cell strings for cell centered grids, and nodes and nodestrings for mesh centered grids. Also includes eools for editing model parameters for Cartesian Finite Difference models.

Scatter. Used for surface modeling with Triangulated Irregular Networks (TINs) or Digital Elevation Maps (DEMSs) used for background data.

Map. Used to build conceptual models using Feature Objects and Conceptual models.

GIS. Used to read in GIS objects directly using ARC Objects. Includes tools to select groups of objects and convert those objects into Feature Objects in the Map module.

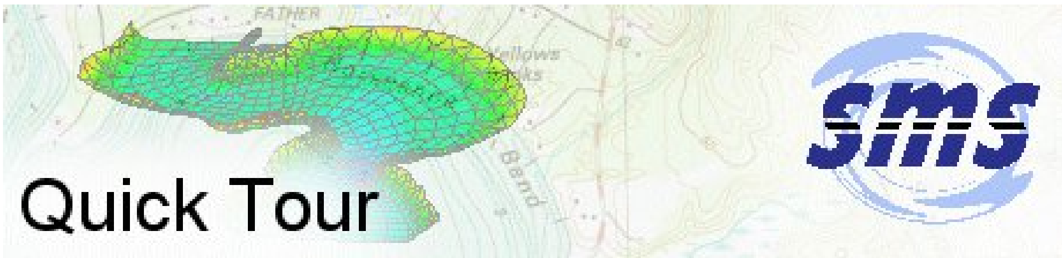
1D River. Used to edit and manage river cross sections along river reaches for 1D hydraulic models.

1D Grid. Used to edit 1D grids used in beach morphology modeling.

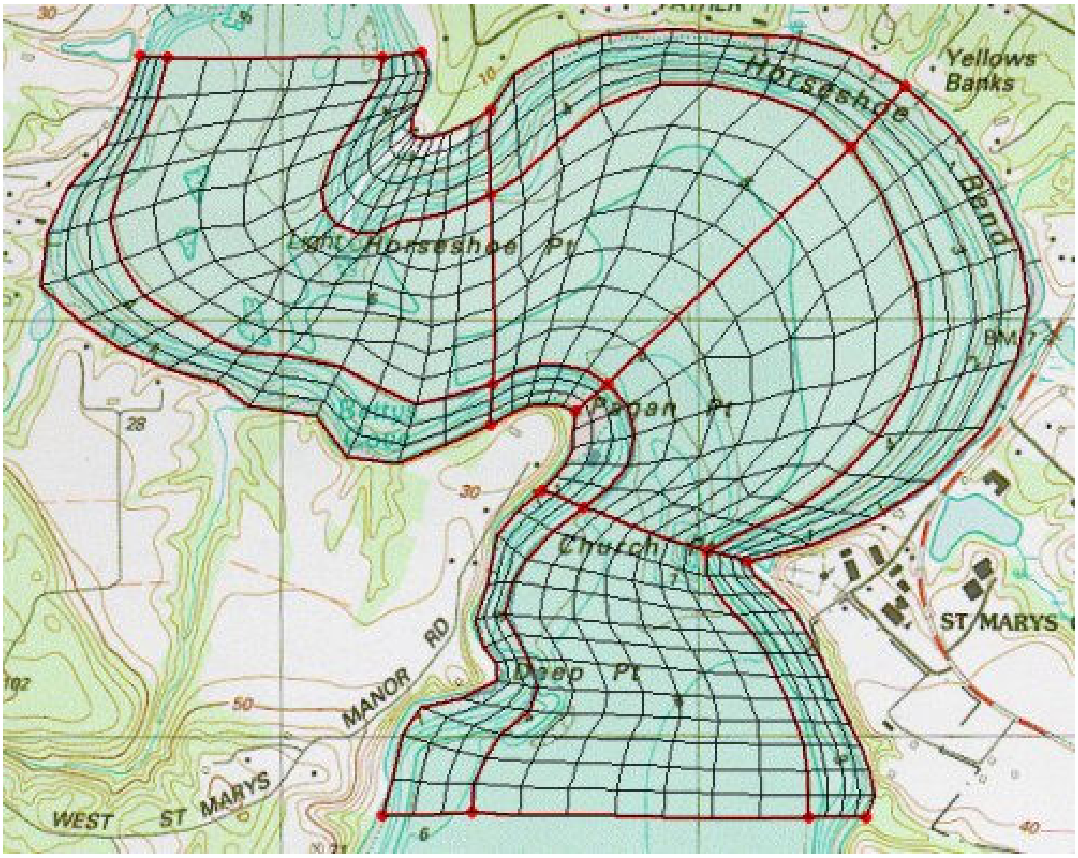
Particle. Used to manage and create Lagrangian particle trace simulations and display their results.

< Previous | Next >

Quick Tour Network Creation



Once the conceptual model is created, a mesh or grid is automatically generated from the conceptual model. The mesh is fit to the model boundary and elements are created according to the assigned element attributes.



< Previous | Next >

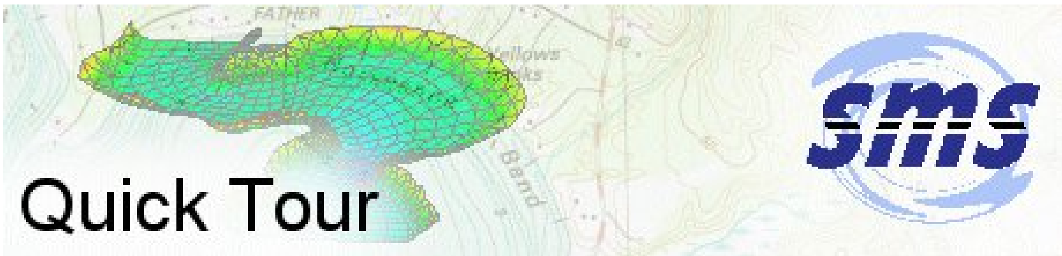
Quick Tour Observation Coverage... to Learn More

To learn how to calibrate models using SMS, complete the following tutorial exercise in the SMS Tutorials document:

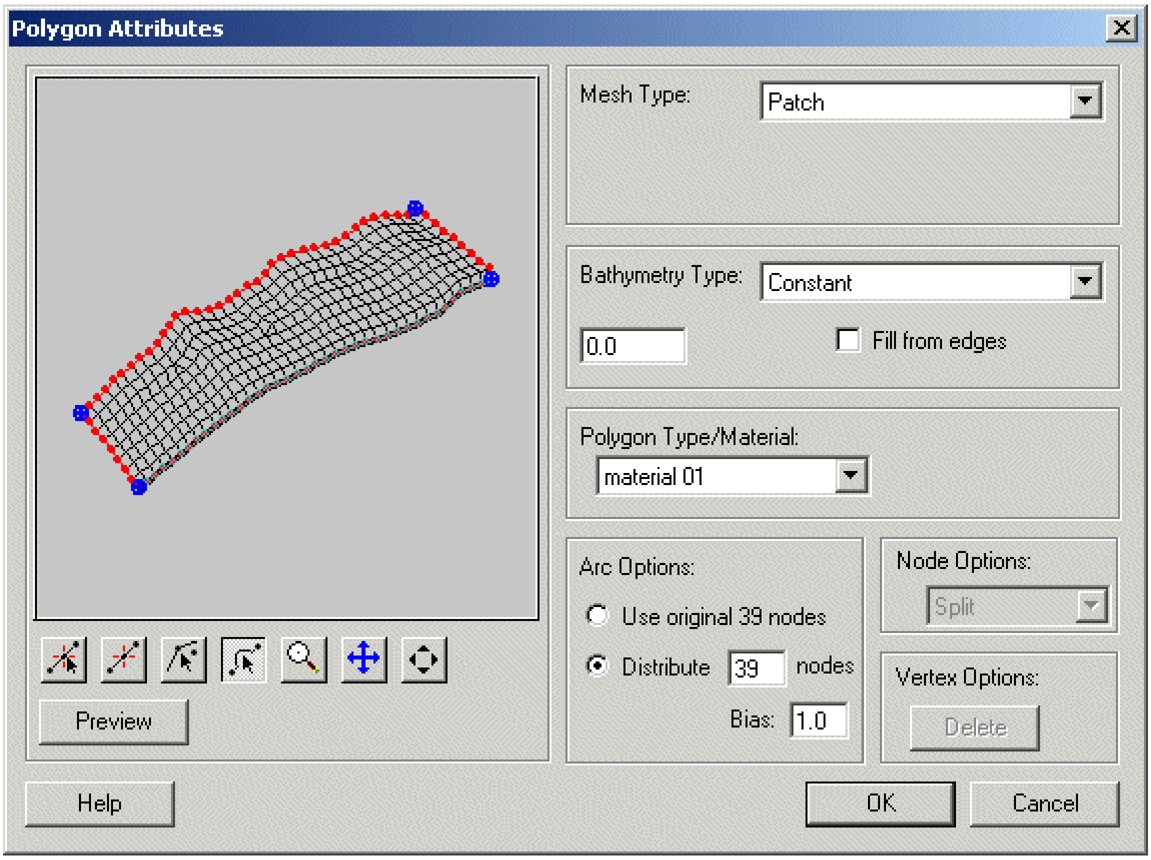
- Observation Coverage

< Previous | Next >

Quick Tour Polygon Attributes

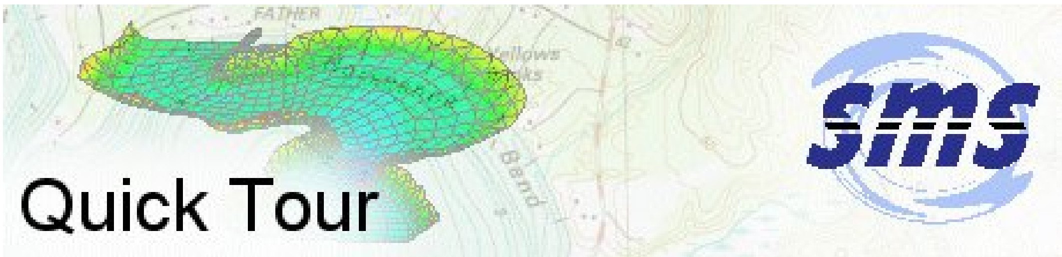


For each polygon, the type of element used to fill that polygon is controlled by the user. SMS supports adaptive tessellation which fills the area with triangles, patches which lines the region with mostly quadrilaterals and density meshing which builds elements based on underlying size functions. All elements in the polygon are assigned the material type for the polygon. All nodes generated in the polygon are assigned bathymetry from the specified source.

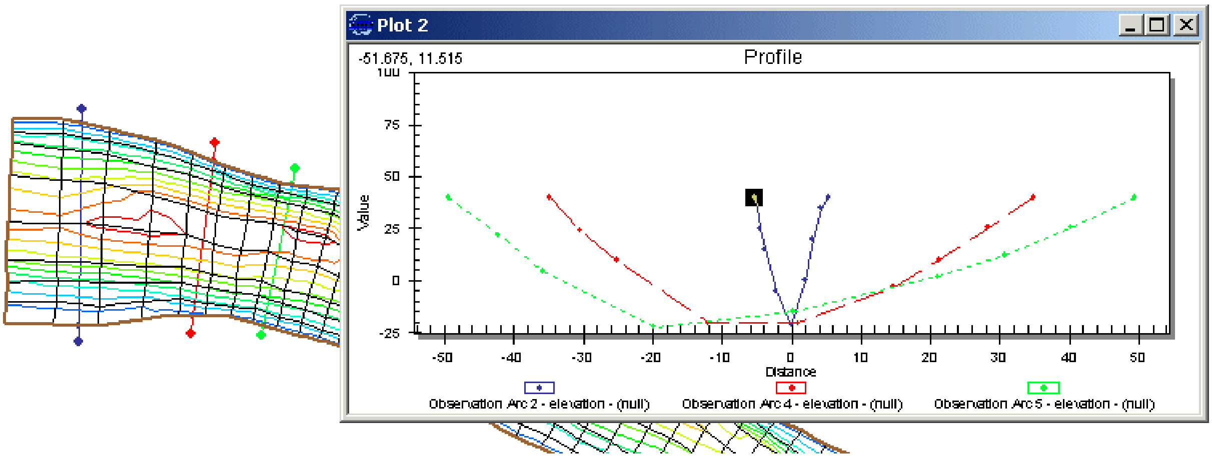


< Previous | Next >

Quick Tour Profiles

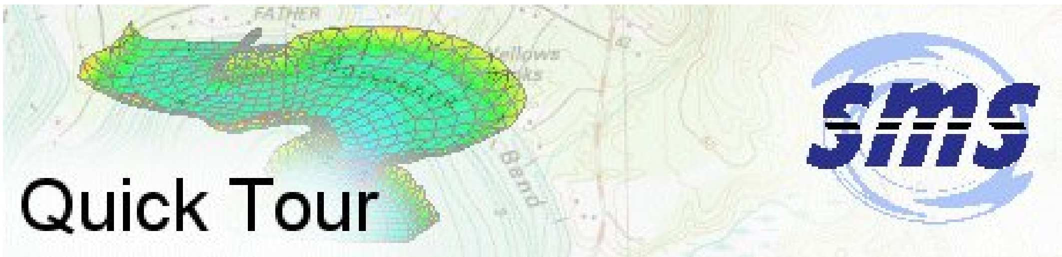


SMS provides tools for plotting a profile along a user specified cross section.



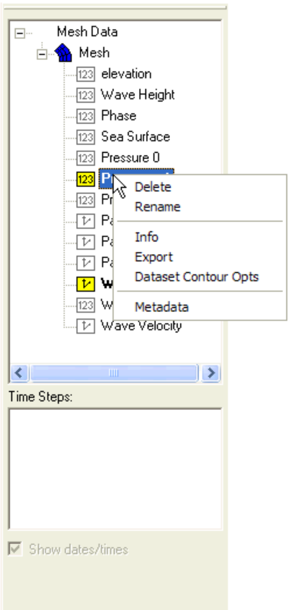
< Previous | Next >

Quick Tour Project Explorer



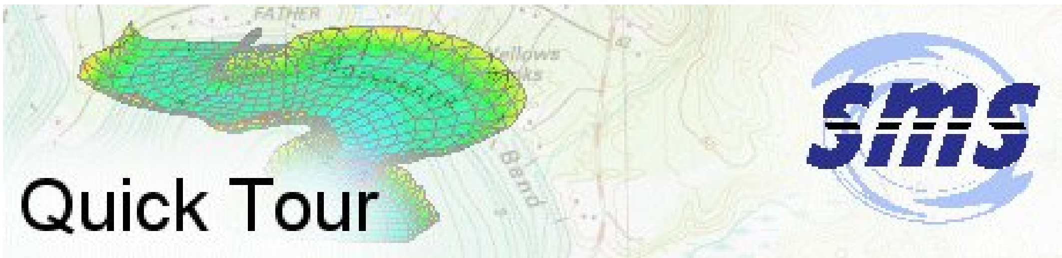
The Project Explorer is a tree representation of all the data in the project. It includes a folder at the base level for type of data. Therefore, if a mesh exists in the simulation, there will be a folder labeled "Mesh Data". If one or more scatter sets exist, there will be a folder labeled "Scattered Data". Inside the folders, SMS will create a tree node for each object of that type. Each object is itself a folder that can contain the functional data sets associated with that object.

The example below shows a mesh named "Mesh" in the Mesh Data portion of the tree with several functions that may be selected. The icon at the left of the function name identifies the function as either scalar or vector. Right clicking on any of the tree objects brings up a menu to edit/select/modify the object or data set.



< Previous | Next >

Quick Tour Scatter Data

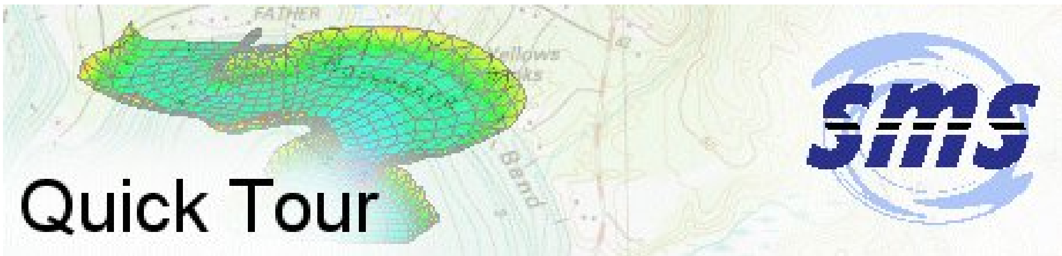


SMS supports a powerful array of options for interpolation in the Scatter Point module. The following interpolation options are supported:

- Linear
- IDW
  - Constant, gradient-plane, and quadratic nodal functions
  - Multiple search/subset schemes
- Natural Neighbor
  - Constant, gradient-plane, and quadratic nodal functions

< Previous | Next >

Quick Tour Support

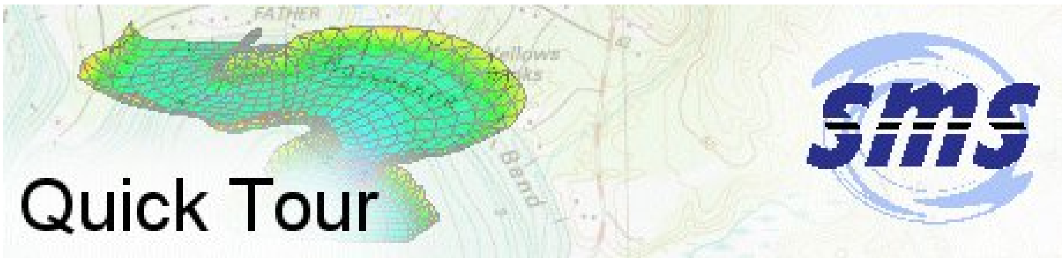


Free technical support is provided for all licensed users of SMS. Regularly scheduled training courses are also available. Contact your SMS vendor for details.

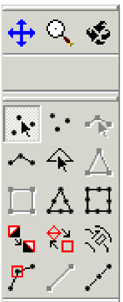
Click here to return to the beginning of the SMS Quick Tour, or here to return to the startup page.

< Previous

Quick Tour Tools



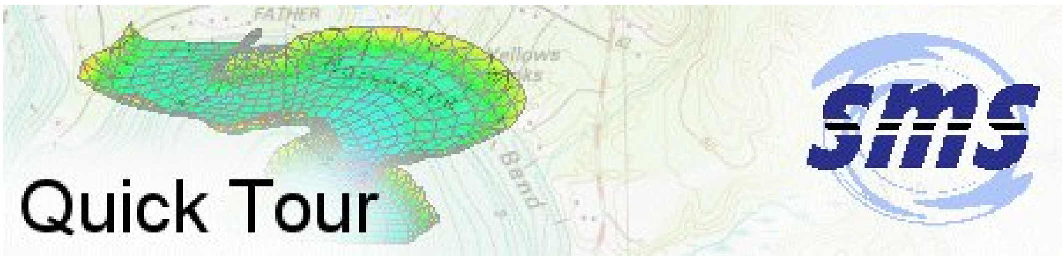
The action that takes place when you click in the graphics window depends on which tool is active. The tools are organized into two groups. The first set of three icons is for manipulating the display. The second set of icons is a group of module-specific tools that are used for creating and editing objects in the graphics window.



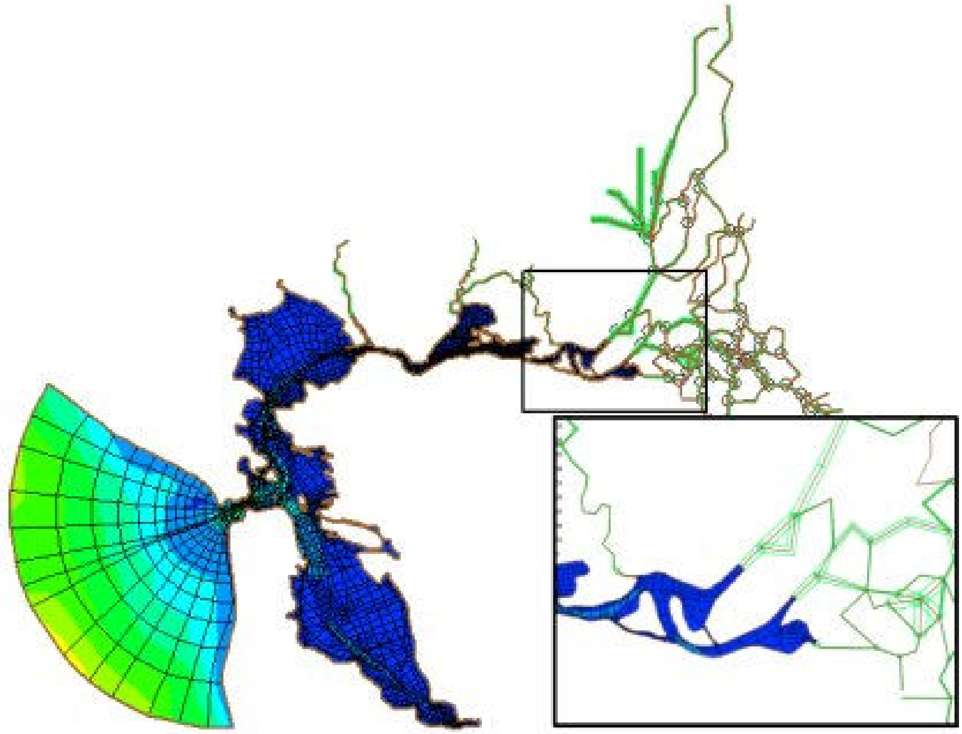
The set of icons shown in this section of the tool palette are a function of the active module. A unique set of tools is provided for each module.

< Previous | Next >

Quick Tour USACE-WES Models



SMS includes interfaces to many of the models supported and used by the United States Army Corps of Engineers Waterways Experiment Station [1]. WES has played an important part in the development and evolution of the tools in SMS. The USACE models with interfaces in SMS include:



2D Finite Element Models

- **CGWAVE** - CGWAVE models harbor response taking into account outside sea state, harbor shape and man-made structures (i.e., piers, breakwaters, naval vessels). It is a forecasting and nowcasting tool used in coastal and military planning and civil engineering.
- **IMS-ADCIRC** - ADCIRC (ADvanced CIRCulation Multi-dimensional Hydrodynamic Model) is a latest-generation multidimensional model based on the solution of the generalized wave equation formulation of the governing equations on a highly flexible unstructured grid.
- **TABS-MD** - TABS is a multidimensional hydrodynamics package. Recently a new version (4.5x) was released for general use by the public. The package includes:
  - **RMA2** - RMA2 is a subcritical, depth averaged model for both steady state and transient hydraulic modeling. General applications require an unstructured finite element network of quadratic elements (6 noded triangles and 8 noded quadrilaterals). Sections of one dimensional entities which simulate trapezoidal channels can be incorporated into the two dimensional mesh.
  - **GFGEN** (Geometry File GENerator) - The models which comprise TABS-MD all utilize a binary representation of the geometry. This binary file is generated from an ASCII file saved by SMS by running the program GFGEN from the RMA2 menu in SMS.
  - **RMA4** - RMA4 is a constituent migration modeling code that has the ability to compute constituent concentrations and dispersion when supplied with a hydrodynamic solution computed by RMA2.
- **IMS-ADCIRC** - ADCIRC (ADvanced CIRCulation Multi-dimensional Hydrodynamic Model) is a latest-generation multidimensional model based on the solution of the generalized wave equation formulation of the governing equations on a highly flexible unstructured grid.

Finite Difference Models

- **BOUSS2D**- BOUSS2D is a comprehensive model for simulating the propagation and tranformation of waves in coastal regions and harbors based on a time-domain solution of Boussinesq-type equations..
- **IMS-M2D**- GHOST models irregular wave propagation in coastal areas. Waves are defined by its directional spectrum.
- **STWAVE** - STWAVE (STeady State Irregular WAVE Model) estimates nearshore wind-wave growth, transformation and propagation

One-Dimensional Model

- **HecRas** - HecRas is designed to predict water surface elevation and velocities for a river network.

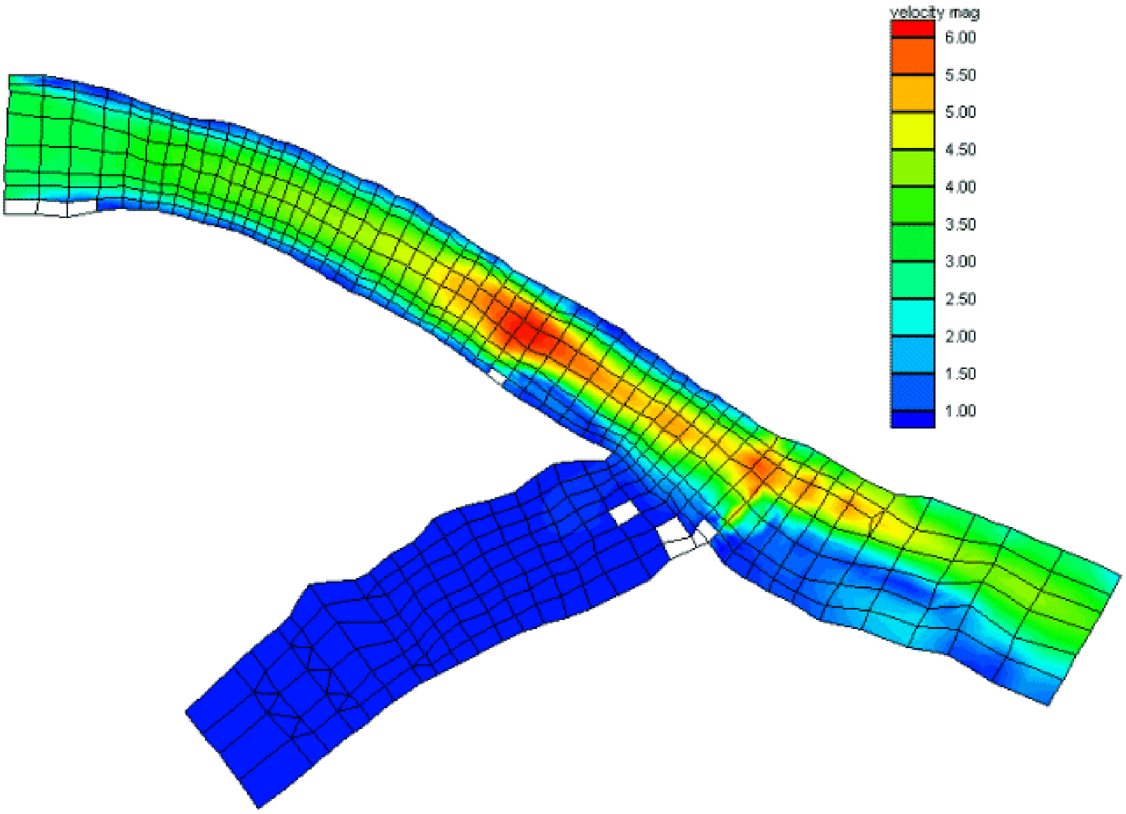
< Previous | Next >

References

[1] http://www.wes.army.mil/

Quick Tour Visualization

SMS provides a powerful suite of tools for generating professional looking quickly and easily. The following figure illustrates a contour plot of velocity magnitude with a user specified color ramp.



< Previous | Next >

7.2. File Support

XMDF

Description

XMDF is a C and Fortran language library providing a standard format for the geometry data storage of river cross-sections, 2D/3D structured grids, 2D/3D unstructured meshes, geometric paths through space, and associated time data. XMDF uses HDF5 for cross-platform data storage and compression.

Version 2.1

- Online Documentation [1]
- XMDF library code [2] (5.9 MB) No XMDF Unix add-on available - Last Revised July 30, 2012
- Change Log [3]

Previous Versions

2.0

- Online Documentation [4]
- XMDF library code [5] (14.6 MB) No XMDF Unix add-on available - Last Revised April 30, 2012

1.9

- Online Documentation [6]
- XMDF library code [7] (6.68 MB) No XMDF Unix add-on available - Last Revised July 12, 2011

1.8

- Online Documentation [8]
- XMDF library code [9] (14.9 MB) No XMDF Unix add-on available - Last Revised April 6, 2011

1.7

- Online Documentation [10]
- XMDF library code [11] (6.52 MB) No XMDF Unix add-on available - Last Revised Jan 19, 2010

1.6

- Online Documentation [12]
- XMDF library code [13] (5.5 MB) No XMDF Unix add-on available - Last Revised Mar 13, 2009

1.4

- Online Documentation [14]
- XMDF library code [15] (5.2 MB) No XMDF Unix add-on available - Last Revised Feb 19, 2008

1.3

- Online Documentation [16]
- XMDF library code [17] (7.8 MB) XMDF Unix add-on [18] (8.4 MB) - Last Revised Oct 4, 2006

1.2

- Online Documentation [19]
- XMDF library code [20] (5.5 MB)

1.1

- Online Documentation [21]
- XMDF library code [22] (2.5 MB)

1.0

- XMDF library code [23] (2.7 MB)

HDF5 File Browsers and Editors

- HDF Explorer [24]
- NCSA HDFView [25]

Back to XMS

References

[1] http://xmdf.aquaveo.com/doc2.1/html/index.html
[2] http://xmdf.aquaveo.com/xmdf_2.1.zip
[3] http://xmdf.aquaveo.com/changes.txt
[4] http://xmdf.aquaveo.com/doc2.0/html/index.html
[5] http://xmdf.aquaveo.com/xmdf_2.0.zip
[6] http://xmdf.aquaveo.com/doc1.9/index.html
[7] http://xmdf.aquaveo.com/xmdf_1.9.zip
[8] http://xmdf.aquaveo.com/doc1.8/index.html
[9] http://xmdf.aquaveo.com/xmdf_1.8.zip
[10] http://xmdf.aquaveo.com/doc1.7/index.html
[11] http://xmdf.aquaveo.com/xmdf_1.7.zip
[12] http://xmdf.aquaveo.com/doc1.6/index.html
[13] http://xmdf.aquaveo.com/xmdf_1.6.zip
[14] http://xmdf.aquaveo.com/doc1.4/index.html
[15] http://xmdf.aquaveo.com/xmdf-1.4.zip
[16] http://xmdf.aquaveo.com/doc1.3/index.html
[17] http://xmdf.aquaveo.com/xmdf-1.3.zip
[18] http://xmdf.aquaveo.com/xmdf-1.3_unix.zip
[19] http://xmdf.aquaveo.com/doc1.2/XMDF.htm
[20] http://xmdf.aquaveo.com/xmdf-lib_1.2.zip
[21] http://xmdf.aquaveo.com/doc1.1/XMDF.htm
[22] http://xmdf.aquaveo.com/xmdf-lib_1.1.zip
[23] http://xmdf.aquaveo.com/xmdf-lib_1.0.zip
[24] http://www.space-research.org/explorer/explorer.htm
[25] http://www.hdf-group.org/hdf-java-html/hdfview/index.html

7.2.a. File Formats

File Formats

Files used in SMS can be grouped into the following categories:

- **Native SMS Files –** Non-model specific files SMS can read and write
- **Non-native SMS Files –** Non-model specific files SMS can read, but must convert and save to a different format
- **Model Input and Output Files –** Files created by numerical models SMS can read.

Related Topics

- File Extensions
- Importing Non-native SMS Files

2D Mesh Files *.2dm

A finite element mesh can be saved in a generic format defined by SMS, called the 2dm format. In addition, the Generic Model interface in the Mesh module uses this format to save a template definition, in addition to model parameter, material property, and boundary condition assignments for a specific simulation. When a *.2dm file is opened, the Current Numerical Model changes to the Generic Mesh Interface. To save a numerical model definition as a template file, you should first set up the template and save a *.2dm file before any nodes are created.

Mesh Cards

Card Type	**MESH2D**
Description	Identifies the file as a 2d mesh file. Must be the first line of the file.
Required	YES

Card Type	**NUM_MATERIALS_PER_ELEM**		
Description	Defines Number of Materials per Element		
Required	YES		
Format	MATERIALS Quantity		
Sample	NUM_MATERIALS_PER_ELEM 2		
Field	**Variable**	**Value**	**Description**
1	Quantity	+ integer	Number of Materials

Nodes

Card Type	**ND**		
Description	Defines the ID and location for each node of the mesh.		
Required	NO		
Format	ND id x y z		
Sample	ND 1 7.75e+005 1.10e+005 5.00e-001		
Field	**Variable**	**Value**	**Description**
1	id	+ integer	The ID of the node.
2-4	x,y,z	± real number	The x, y, and z coordinates of the point.

Linear Elements

Card Type	**E2L**		
Description	Identifies a 2-noded linear element.		
Required	NO		
Format	E2L id n_1 n_2 matid		
Sample	E2L 1 1 2 1		
Field	**Variable**	**Value**	**Description**
1	id	+ integer	The ID of the element.
2 - 3	n_1 - n_2	+ integer	The ID's of nodes in the element.
4	matid	+ integer	The ID of the material assigned to the element.

Card Type	**E3L**		
Description	Identifies a 3-noded linear element.		
Required	NO		
Format	E3L id n_1 n_2 n_3 matid		
Sample	E3L 1 1 2 3 1		
Field	**Variable**	**Value**	**Description**
1	id	+ integer	The ID of the element.
2 - 4	n_1 - n_3	+ integer	The ID's of nodes in the element.
5	matid	+ integer	The ID of the material assigned to the element.

Triangular Elements

Card Type	E3T		
Description	Identifies a 3-noded triangular element.		
Required	NO		
Format	E3T id n_1 n_2 n_3 matid		
Sample	E3T 1 1 2 3 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 4	n_1 - n_3	+ integer	The ID's of nodes in the element.
5	matid	+ integer	The ID of the material assigned to the element.

Card Type	E6T		
Description	Identifies a 6-noded triangular element.		
Required	NO		
Format	E6T id n_1 n_2 n_3 n_4 n_5 n_6 matid		
Sample	E6T 1 1 2 3 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 7	n_1 - n_6	+ integer	The ID's of nodes in the element.
8	matid	+ integer	The ID of the material assigned to the element.

Quadrilateral Elements

Card Type	E4Q		
Description	Identifies a 4-noded quadrilateral element.		
Required	NO		
Format	E4Q id n_1 n_2 n_3 n_4 matid		
Sample	E4Q 1 1 2 3 4 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 5	n_1 - n_4	+ integer	The ID's of nodes in the element.
6	matid	+ integer	The ID of the material assigned to the element.

Card Type	E8Q		
Description	Identifies an 8-noded quadrilateral element.		
Required	NO		
Format	E8Q id n_1 n_2 n_3 n_4 n_5 n_6 n_7 n_8 matid		
Sample	E8Q 1 1 2 3 4 5 6 7 8 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 9	n_1 - n_8	+ integer	The ID's of nodes in the element.
10	matid	+ integer	The ID of the material assigned to the element.

Card Type	E9Q		
Description	Identifies an 9-noded quadrilateral element.		
Required	NO		
Format	E9Q id n_1 n_2 n_3 n_4 n_5 n_6 n_7 n_8 n_9 matid		
Sample	E9Q 1 1 2 3 4 5 6 7 8 9 1		
Field	Variable	Value	Description
1	id	+ integer	The ID of the element.
2 - 10	n_1 - n_9	+ integer	The ID's of nodes in the element.
11	matid	+ integer	The ID of the material assigned to the element.

Nodestrings

Card Type	NS		
Description	Identifies a nodestring.		
Required	NO		
Format	NS n_1 n_2 n_3 ... - n_n (number of nodes in nodestring)		
Sample	NS 1 3 10 15 6 -2 NS 126 127 128 129 173 -194 NS 1006 988 987 989 968 948 931 930 929 906 NS -904 NS 720 701 699 686 680 664 649 648 647 640 NS -621		
Field	Variable	Value	Description
n_1 - n_n (number of nodes in nodestring)	n_x	+ integer	The ID's of the nodes in the nodestring. The last node id is written as a negative number, thus signaling the nodestring's end. Multiple NS cards can be used on consecutive lines for a single nodestring.

Model Parameter Definition Cards

The model control parameters, boundary conditions and material options available for a specific model are defined in the generic model parameter definition section of the 2D mesh file. The generic model parameter definition section is begun by specifying the **BEGPARAMDEF** card and ended with the **ENDPARAMDEF** card.

Starting with SMS version 11.0, all of the parameter cards follow a similar pattern and the available options for several of the types of parameters were increased. The pattern is similar for Global Parameters, **GP**, Bondary Conditions **BC**, and Materials **MAT**. More information can be found in each individual card.

Global Parameters	Boundary Conditions	Materials	Description
GP	BC	MAT	Name,Id
GP_DEF	BC_DEF	MAT_DEF	definitions
GP_OPTS	BC_OPTS	MAT_OPTS	options
GP_VAL	BC_VAL	MAT_VAL	values
(parent, so it has no dependencies)	BC_DEP	MAT_DEP	dependencies

Card Type	**BEGPARAMDEF**
Description	Identifies the beginning of the model parameter definition section of the 2D Mesh File.
Required	Required if model parameters are to be defined.

Card Type	**ENDPARAMDEF**
Description	Identifies the end of the model parameter definition section of the 2D Mesh File.
Required	Required if model parameters are to be defined.

Global Parameters

Card Type	**GM**		
Description	Identifies the model name.		
Required	NO		
Format	GM name		
Sample	GM "Gen2DM"		
Field	**Variable**	**Value**	**Description**
1	name	string	Model name.

Card Type	**SI**		
Description	Identifies the model units.		
Required	NO		
Format	SI val		
Sample	SI 0		
Field	**Variable**	**Value**	**Description**
1	val	boolean	Enter 1 for Meters, or 0 for U.S. Survey Feet or International Feet.

Card Type	**DY**		
Description	Identifies whether the model is dynamic or steady state.		
Required	NO		
Format	DY val		
Sample	DY 1		
Field	**Variable**	**Value**	**Description**
1	val	boolean	• Enter 0 for steady state. • Enter 1 for dynamic.

Card Type	**TU**		
Description	Identifies the model time units.		
Required	NO		
Format	TU val		
Sample	TU seconds		
Field	**Variable**	**Value**	**Description**
1	val	string	A string value describing the model units (days, hours, minutes, seconds, etc.).

Card Type	**TD**		
Description	Identifies the model time step and total simulation time.		
Required	NO		
Format	TD time_step total_time		
Sample	TD 20 1000		
Field	**Variable**	**Value**	**Description**
1	time_step	+ real number	Time step (for dynamic simulations).
1	total_time	+ real number	Total simulation run time (for dynamic simulations).

Card Type	**KEY**		
Description	Identifies the key to unlock and edit the model definition inside of the SMS interface.		
Required	NO		
Format	KEY key		
Sample	KEY "sms-gen2dm"		
Field	**Variable**	**Value**	**Description**
1	key	string	Case sensitive key to unlock and edit the model definition.

Card Type	• **DISP_OPTS entity** • **DISP_OPTS inactive** • **DISP_OPTS multiple**		
Description	How a specific display option to the user.. colors, line thickness, etc. • **entity –** the main attribute (will either be the node, element, or nodestring display option) • **inactive –** display option for inactive or unassigned • **multiple –** display option for multiple assigned		
Required	NO		
Format	DISP_OPTS EntityId Red Green Blue Display Pattern Width Style		
Sample	DISP_OPTS entity 1 0 0 0 1 0 1 0		
Sample	DISP_OPTS multiple 1 0 0 0 1 0 1 0		
Field	**Variable**	**Value**	**Description**
1	EnitityId	integer (0-2) • 0 = node • 1 = nodestring • 2 = element	id of the group it belongs too.
2	Red	integer (0-255)	red pixels
3	Green	integer (0-255)	green pixels
4	Blue	integer (0-255)	blue pixels
5	Display	boolean (0,1)	turned on = 1, turned off = 0
6	Pattern	integer	display pattern
7	Width	integer	width
8	Style	integer	style pattern

Global Parameter Assignment Cards

Card Type	GP		
Description	Defines a Global Parameter Group		
Required	NO		
Format	GP Id Name Active		
Sample	GP 1 "Hydro" 1		
Field	Variable	Value	Description
1	+ integer	id	
2	name	string	name
3	active	boolean	• 0 = inactive • 1 = active

Card Type	GP_DEF		
Description	Global Parameter Defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : GP_DEF GroupId ParamId Name Type Default Min Max curve: GP_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: GP_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_titleGP param_name val		
Sample	Values depend upon type (see format). bool, integer, double, text or options : GP_DEF 1 1 "manning n" 1 0 0 10 curve: GP_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: GP_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	Variable	Value	Description
1	group Id	+ integer	id of the Global Parameter Group that it belongs too
2	param Id	string	its id
3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	• default • or x_axis_title (curve)	• type specific () • string	• default value • x-axis title when bringing up curve editor
6	• min or • y_axis_title (curve)	• type specific (min) • string	• minimum value • y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either • "FLOAT" • "CURVE"	The default version float or integer that is seen by the user
9 (only float/curve)	x_axis_title	string	x-axis title when bringing up curve editor

10 (only float/curve)	y_axis_title	string	y-axis title when bringing up curve editor

Card Type	GP_VAL		
Description	Global Parameters values		
Required	NO		
Format	GP_VAL GroupId ParamId (CURVE or FLOAT only if type is float/curve) Value		
Sample	GP_VAL 1 1 30.23		
Sample	GP_VAL 1 2 "Manning"		
Field	**Variable**	**Value**	**Description**
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id
3	Value	varies depending on type	value

Card Type	GP_DEP		
Description	Global Parameters dependencies		
Required	NO		
Format	GP_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	GP_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	**Variable**	**Value**	**Description**
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id
3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	Parent Active	boolean	• 0 = inactive • 1 = active
6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Boundary Condition Definition Cards

Card Type	BCPGC		
Description	Defines whether or not to allow boundary condition / parameter group correlation.		
Required	NO		
Format	BCPGC val		
Sample	BCPGC 1		
Field	**Variable**	**Value**	**Description**
1	val	boolean	• 0 = Do not allow boundary condition / parameter group correlation. • 1 = Allow boundary condition / parameter group correlation.

Card Type	BEDISP		
Description	Defines inactive boundary condition display options and boundary condition label options.		
Required	NO		
Format	BEDISP entity_ID font_red font_green font_blue label_on label_vals_on inactive_size inactive_style inactive_red inactive_green inactive_blue inactive_on		
Sample	BEDISP 0 2 0 0 0 1 1 1 255 128 255 1		
Field	**Variable**	**Value**	**Description**
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element
2	font_red	integer	0 - 255, Red component of RGB triplet defining boundary condition font color.
3	font_green	integer	0 - 255, Green component of RGB triplet defining boundary condition font color.
4	font_blue	integer	0 - 255, Blue component of RGB triplet defining boundary condition font color.
5	label_on	boolean	• 0 = Do not display boundary condition labels. • 1 = Display boundary condition labels.
6	label_vals_on	boolean	• 0 = Do not display boundary condition values in boundary condition labels. • 1 = Display boundary condition values in boundary condition labels.
7	inactive_size	integer	• 1 - 63, Default size for inactive boundary condition node symbols. • 1 - 50, Default size for inactive boundary condition element or nodestring symbols.

8	inactive_style	integer	• Default style for inactive boundary condition symbols.
			• For nodes
			  • 1 = Filled square
			  • 2 = Hollow square
			  • 3 = Filled circle
			  • 4 = Hollow circle
			  • 5 = Filled triangle
			  • 6 = Hollow triangle
			  • 7 = Filled diamond
			  • 8 = Hollow diamond
			  • 9 = Cross
			  • 10 = X
			  • 11 = Survey marker
			• For elements or nodestrings:
			  • 0 = solid line
			  • 1 = dashed line

9	inactive_red	integer	0 - 255, Red component of RGB triplet defining inactive boundary condition font color.
10	inactive_green	integer	0 - 255, Green component of RGB triplet defining inactive boundary condition font color.
11	inactive_blue	integer	0 - 255, Blue component of RGB triplet defining inactive boundary condition font color.
12	inactive_on	boolean	• 0 = Do not display inactive boundary conditions. • 1 = Display inactive boundary conditions.

Card Type	**BEFONT**	Single integer method.	
Description	Defines boundary condition label font attributes.		
Required	NO		
Format	BEFONT entity_id font_size		
Sample	BEFONT 1 1		
Field	**Variable**	**Value**	**Description**
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element
2	font_size	integer	• 1 = Large font size. • 2 = Small font size.

Card Type	**BEFONT**	Multiple value method.	
Description	Defines boundary condition label font attributes.		
Required	NO		
Format	BEFONT entity_id height width escapement orientation weight italic underline strikeout charSet precision clipPrecision quality pitchAndFamily faceName		
Sample	BEFONT 1 1		
Field	**Variable**	**Value**	**Description**
1	entity_ID	integer	• 0 = node • 1 = nodestring • 2 = element
2	height	integer	Font height.

3	width	integer	Font width.
4	escapement	integer	Font escapement.
5	orientation	integer	Font orientation.
6	weight	integer	Font weight.
7	italic	integer	Font italic.
8	underline	integer	Font underline.
9	strikeout	integer	Font strikeout.
10	charSet	integer	Font character set.
11	precision	integer	Font precision.
12	clipPrecision	integer	Font clip precision.
13	quality	integer	Font quality.
14	pitchAndFamily	integer	Font pitch and family.
15	faceName	string	Font face name.

Card Type	**BC_DISP_OPTS**		
Description	Boundary Condition display options (how the boundary conditions are going to be displayed to the user)		
Required	NO		
Format	BC_DISP_OPTS GroupId Red Green Blue Display Pattern Width Style		
Sample	BC_DISP_OPTS 1 0 0 0 0 0 0 0 0		
Field	**Variable**	**Value**	**Description**
1	Group Id	integer	• 0 = node • 1 = nodestring • 2 = element
2	Red	integer (0-255)	red pixels
3	Green	integer (0-255)	green pixels
4	Blue	integer (0-255)	blue pixels
5	Display	boolean (0,1)	turned on = 1, turned off = 0
6	Pattern	integer	displayed pattern
7	Width	integer	width
8	Style	integer	style pattern

Card Type	**BEG2DMBC**
Description	Identifies the beginning of the boundary condition assignment section of the 2D Mesh File.
Required	Required if boundary conditions are assigned.

Card Type	END2DMBC
Description	Identifies the end of the boundary condition assignment section of the 2D Mesh File.
Required	Required if boundary conditions are assigned.

Boundary Condition Assignment Cards

The following cards are used to assign values to the boundary conditions defined in using **Boundary Condition Definition Cards**:

Card Type	BC		
Description	Defines input values required for a boundary condition.		
Required	NO		
Format	BC EntityId Name Id 0 LegalOnInterior ParamGroupCorrelation		
Sample	• BC 1 "Flow rate (cfs)" 1 0 1 "(none)" • BC 1 "Water temperature (F)" 2 0 "Hydrodynamic" • BC 1 "Flow rate(cfs)" 3 0 1 "(none)"		
Field	**Variable**	**Value**	**Description**
1	entity id	integer	Entity id that the bc belongs to (NODE = 0, NDSTR =1, ELEM = 2
2	name	string	Name of value to be specified.
3	id	integer	The boundary conditions id.
4	filler	0	Always 0
5	legalOnInterior	boolean	• 0 = boundary condition is not legal on the interior of the mesh. • 1 = boundary condition is legal on the interior of the mesh.
6	paramGroupCorrelation	string	Name of the parameter group (defined using the GP card) with which the boundary condition is correlated. If the boundary condition is not correlated with a parameter group, "none" should be specified.

Card Type	BC_DEF		
Description	Boundary Condition Parameter Defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : BC_DEF GroupId ParamId Name Type Default Min Max curve: BC_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: BC_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_title		
Sample	Values depend upon type (see format). bool, integer, double, text or options : BC_DEF 1 1 "manning n" 1 0 0 10 curve: BC_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: BC_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	**Variable**	**Value**	**Description**
1	group Id	+ integer	id of the boundary condition group that it belongs too
2	param Id	string	its id

3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	default or x_axis_title (curve)	type specific string	default value x-axis title when bringing up curve editor
6	min or y_axis_title (curve)	type specific (min) string	minimum value y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either • "FLOAT" • "CURVE"	The default version float or integer that is seen by the user
9 (only float/curve	x_axis_title	string	x-axis title when bringing up curve editor
10 (only float/curve)	y_axis_title	string	y-axis title when bringing up curve editor

Card Type	**BC_OPTS**		
Description	Boundary Condition options		
Required	NO		
Format	BC_OPTS GroupId, ParamId, Values		
Sample	BC_OPTS 1 5 "a" "b" "c"		
Field	**Variable**	**Value**	**Description**
1	group Id	+ integer	id of the boundary condition group that it belongs too
2	param Id	string	its id
3	values	string	

Card Type	**BC_VAL**		
Description	Boundary Condition values		
Required	NO		
Format	BC_VAL N, E, S (Node, Elem or nodeString) Node or Elem Id, GroupId, ParamId, (CURVE or FLOAT only if type is float/curve) Value		
Sample	BC_VAL S 1 1 2 2		
Sample	BC_VAL S 1 1 2 CURVE 2		
Sample	BC_VAL S 1 1 2 FLOAT 7.675		
Field	**Variable**	**Value**	**Description**
1	N, E or S	+ integer	Node, Elem or Nodestring
2	Node, Elem, or Nodestring Id	+ integer	id of the node/element
3	group Id	+ integer	id of the group it belongs too
4	param Id	+ integer	id of the parameter it belongs too
5	value	varies depending on type	value

Card Type	BC_DEP		
Description	Boundary Condition dependencies		
Required	NO		
Format	BC_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	BC_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	**Variable**	**Value**	**Description**
1	Group Id	+ integer	id of the Global Parameter Group that it belongs too
2	Param Id	+ integer	its id
3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	Parent Active	boolean	• 0 = inactive • 1 = active
6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Material Properties Cards

Card Type	MAT_MULTI		
Description	Is Material Assignment Multiple		
Required	NO		
Format	MAT_MULTI Assigned		
Sample	MAT_MULTI 1		
Sample	MAT_MULTI 0		
Field	**Variable**	**Value**	**Description**
1	assigned	boolean	0- Single, 1 - Multiple

Card Type	MAT_PARAMS		
Description	Material Parameters Assigned		
Required	NOT NEEDED IF MAT_MULTI 0		
Format	MAT_PARAMS MaterialId, GroupId (1 or More)		
Sample	MAT_PARAMS 1 2 3 5		
Sample	MAT_PARAMS 1 1		
Field	**Variable**	**Value**	**Description**
1	Material Id	+ integer	id of the material
2	Group id	+ integer	id of the assigned group

Material Properties Assignment Cards

Card Type	MAT		
Description	Defines the assigned material properties.		
Required	NO		
Format	MAT ID "name"		
Sample	MAT 1 "Clay"		
Field	**Variable**	**Value**	**Description**
1	mat_ID	+ integer	Material id (sequentially numbered, starting at 1). The disabled material is always id 0 and does not need to be specified in the 2dm file.
2	name	string	The name of the material. Should be unique.

Card Type	MAT_DEF		
Description	Material Parameters defaults		
Required	NO		
Format	The format will depend up the type (field 4). Fields 5, 6, and 7 will be impacted by the choice of field 4. Note that the min and max information is only applicable to integer and double types. bool, integer, double, text, or options : MAT_DEF GroupId ParamId Name Type Default Min Max curve: MAT_DEF GroupId ParamId Name Type x_axis_title y_axis_title Float/Curve: MAT_DEF GroupId ParamId Name Type Float_Default Float_Min Float_Max DefaultFloatOrCurve x_axis_title y_axis_title		
Sample	Values depend upon type (see format). bool, integer, double, text or options : MAT_DEF 1 1 "manning n" 1 0 0 10 curve: MAT_DEF 1 1 "manning n" 5 "x-axis" "y-axis" Float/Curve: MAT_DEF 1 1 "manning n" 6 0.2 0.0 1.0 CURVE "x-axis" "y-axis"		
Field	**Variable**	**Value**	**Description**
1	group id	+ integer	id of the Material Group that it belongs too.
2	param id	string	its id
3	name	string	its name
4	type	int	0-Bool, 1-Integer, 2-Double, 3-Text, 4-Options, 5-Curve, 6-Float/Curve
5	default or x-axis_title (curve)	type specific string	default value x-axis title when bringing up curve editor
6	min or y-axis_title (curve)	type specific (min) string	minimum value y-axis title when bringing up curve editor
7	max	type specific (max)	max value
8 (only float/curve)	DefaultFloatOrCurve	string either • "FLOAT" • "CURVE"	The default version float or integer that is seen by the user
9 (only float/curve)	x_axis_title	string	x-axis title when bringing up curve editor
10 (only float/curve)	y_axis_title	string	y-axis title when bringing up curve editor

Card Type	MAT_OPTS		
Description	Material options		
Required	NO		
Format	MAT_OPTS GroupId, ParamId, Values		
Sample	MAT_OPTS 1 5 "a" "b" "c"		
Field	**Variable**	**Value**	**Description**
1	group id	+ integer	id of the Boundary Condition Group that it belongs too
2	param id	+ integer	its id
3	values	string	

Card Type	MAT_VALS		
Description	Material values		
Required	NO		
Format	MAT_VAL MaterialId, GroupId, ParamId, (CURVE or FLOAT only if type is float/curve) Value		
Sample	MAT_VAL S 1 1 2 2		
Sample	MAT_VAL 2 1 8 VALUE 8.8888		
Sample	MAT_VAL 2 1 8 CURVE 1		
Field	**Variable**	**Value**	**Description**
1	material id	+ integer	id of the material
2	group id	+ integer	id of the group it belongs too
3	param id	string	id of the parameter it belongs too
4	VALUE or CURVE	string	Only if type is float/curve
5	value	varies depending on type	value

Card Type	MAT_DEP		
Description	Material dependencies		
Required	NO		
Format	MAT_DEP GroupId ParamId Type Parent ParentActive Opts OptsValue		
Sample	MAT_DEP 1 7 "PARENT_SELF" "Friction type" 0 "Manning" 0 "Chezy" 1		
Field	**Variable**	**Value**	**Description**
1	GroupId	+ integer	id of the Global Parameter Group that it belongs too
2	ParamId	+ integer	its id
3	Type	string valid values are: • PARENT_UNASSGINED • PARENT_NONE • PARENT_LOCAL • PARENT_GLOBAL • PARENT_SELF	• parent not assigned • no parent • parent is in the same group id • parent is from global group or GP • parent
4	Parent	+ integer	parent name
5	ParentActive	boolean	• 0 = inactive • 1 = active

6	Opts	string	name of the option
7	OptsValue	boolean	whether this option is turned on/off

Time Series Data Cards

Card Type	**BEGCURVE Version: version**		
Description	Identifies the beginning of the time series data section of the 2D Mesh File.		
Required	NO		
Format	BEGCURVE		
Sample	BCE 2 2 300		
Field	**Variable**	**Value**	**Description**
1	version	+ integer	Used to version file format for the curve portion of the file.
Note: Version 1 writes the time series data using the XY Series File, XYS Format within the 2D Mesh File. See XY Series Files (*.xys) for a description of the cards used to define the time series data.			

Card Type	**ENDCURVE**
Description	Identifies the end of the time series data section of the 2D Mesh File.
Required	NO

Obsolete Cards

The following cards are no longer used in SMS:

PG, PD, PO, GG, GP, BD, BV, MD, MV, BCE, BCN, BCS, TIME, BEDISP

Changes from 10.1 to 11.0

New Cards:

- NUM_MATERIALS_PER_ELEM
- GP replaces PG
- GP_DEF replaces PD
- GP_VAL
- GP_DEP
- BC replaces BD
- BC_DEF replaces BV card
- BC_OPTS replaces PO card
- BC_VAL replaces BC card
- BC_DEP
- BC_DISP_OPTS replaces bedisp
- MAT_MULTI
- MAT_PARAMS
- MAT_DEF replaces MV card
- MAT_OPTS
- MAT_VAL

- MAT_DEP
- DISP_OPTS entity
- DISP_OPTS inactive
- DISP_OPTS multiple

Sample File 1

```
BEGCURVE Version: 1
XYS 1 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 3 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
```

```
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 5 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
```

```
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 7 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 9 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
```

```
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
XYS 11 29 "new_series"
0.0 3000.0
1.5 3000.0
2.5 3050.0
3.0 3150.0
3.5 3300.0
4.0 3500.0
4.5 3700.0
5.0 3950.0
5.5 4150.0
6.0 4350.0
6.5 4550.0
7.0 4700.0
7.45 4825.0
8.0 4925.0
8.5 4975.0
9.0 5000.0
9.5 4975.0
10.0 4800.0
10.5 4500.0
11.0 4250.0
11.5 4000.0
12.0 3750.0
13.0 3500.0
14.0 3350.0
15.5 3200.0
17.5 3100.0
19.5 3050.0
22.0 3000.0
25.0 3000.0
```

```
XYS 13 8 "new_series"
0.0 237.35
24.0 137.9
48.0 1347.4
72.0 351.05
96.0 1465.25
120.0 1247.1
144.0 847.35
168.0 731.75
XYS 14 8 "Curve"
0.0 1600.7
24.0 700.85
48.0 1353.7
72.0 712.25
96.0 866.6
120.0 1626.35
144.0 567.6
168.0 980.55
XYS 15 8 "Curve"
0.0 1240.7
24.0 772.25
48.0 741.3
72.0 908.45
96.0 599.45
120.0 522.8
144.0 946.8
168.0 170.4
XYS 16 8 "Curve"
0.0 1252.55
24.0 1467.95
48.0 876.25
72.0 250.2
96.0 479.95
120.0 981.65
144.0 1432.4
168.0 1382.8
XYS 17 8 "Curve"
0.0 1507.65
24.0 202.6
48.0 905.3
72.0 1051.45
96.0 434.95
120.0 267.5
144.0 547.95
168.0 349.2
ENDCURVE
```

Sample File 2

Sample files are available in the SMS tutorials in the Generic Mesh Model tutorial under the models section.

```
MESH2D
E3T      1      4      1      3      2
E3T      2      2      5      6      2
E4Q      3      7      8      5      2      2
.
.
.
E4Q   1543    205   1226   1225   1221      2
E4Q   1544    191   1222   1226    189      1
E3T   1545    205    189   1226      2
ND       1 -7.62907961e+001 4.00243909e+001 8.41808447e+001
ND       2 -7.62907174e+001 4.00219296e+001 8.36614138e+001
ND       3 -7.62907700e+001 4.00238340e+001 7.32122342e+001
.
.
.
ND    1222 -7.62811008e+001 4.00272795e+001 7.28898113e+001
ND    1225 -7.62814608e+001 4.00273631e+001 7.29479847e+001
ND    1226 -7.62812859e+001 4.00271526e+001 7.41231480e+001
NS       1      3     10     15      6     -2
NS     126    127    128    129    173   -194
NS    1006    988    987    989    968    948    931    930    929
906
NS    -904
NS     720    701    699    686    680    664    649    648    647
640
NS    -621
BEGPARAMDEF
GM  "Gen2DM"
SI  0
DY  1
TU  "minutes"
TD  20  1000
KEY  "sms-gen2dm"
PG  "Hydrodynamic" 1
PD  "Time interval" 1 20 0 2147483647
PD  "Velocity max (ft/sec)" 2 75 0 100
PD  "H min" 2 0.25 0 1.79769e+308
PD  "A min" 2 1 1e-015 1.79769e+308
PD  "Check for dry elements" 0 1
PD  "Element style" 3 "quadratic"
PD  "Critical scour velocity" 4 "2.0 ft/sec"
PO  "0.8 ft/sec" "2.0 ft/sec" "2.6 ft/sec"
PG  "Sediment transport" 0
```

```
PD  "Time interval" 1 10 0 2147483647
PD  "Source X postion" 2 0 -1.79769e+308 1.79769e+308
PD  "Source Y position" 2 0 -1.79769e+308 1.79769e+308
PD  "Source elevation" 2 0 -1.79769e+308 1.79769e+308
PD  "Parcel mass (slug)" 2 0.5 0.0001 1.79769e+308
PD  "Particle mass (slug)" 2 0.003 0.0001 1.79769e+308
PD  "Particle size (in)" 2 0.05 0 1.79769e+308
PD  "Deviation" 2 0 -1.79769e+308 1.79769e+308
PD  "Average density (slug/ft^3)" 2 3 1.5 6
NUME  3
BCPGC  1
BEDISP  0 2 0 0 0 1 1 1 255 128 255 1
BEFONT  0 1
BD  0 "Water sink/source" 2 3 "Flow rate (cfs)" "Water temperature (F)"
 "Flow rate(cfs)" 1 "(none)"
BV  "Flow rate (cfs)" 0 -1.79769e+308 1.79769e+308
BV  "Water temperature (F)" 65 32.5 100
BV  "Flow rate(cfs)" 0 0 1.79769e+308
BCDISP  0 2 10 1 0 255 255 1
BD  0 "Ceiling (pressure flow)" 1 1 "Ceiling (ft above sea level)" 0
"(none)"
BV  "Ceiling (ft above sea level)" 0 -1.79769e+308 1.79769e+308
BCDISP  0 1 3 1 128 128 255 1
BD  0 "Water surface observation gauge" 3 0 1 "(none)"
BCDISP  0 3 3 1 255 128 128 1
BEDISP  1 0 0 255 1 1 1 0 255 128 0 1
BEFONT  1 1
BD  1 "Water surface" 1 3 "Elevation" "Essential/Natural factor" "Vary
along nodestring factor" 0 "(none)"
BV  "Elevation" 0 -1.79769e+308 1.79769e+308
BV  "Essential/Natural factor" 0 0 1
BV  "Vary along nodestring factor" 1 0 10
BCDISP  1 1 5 0 255 0 0 1
BD  1 "Flow" 2 1 "Flow rate (cfs)" 0 "(none)"
BV  "Flow rate (cfs)" 0 0 1.79769e+308
BCDISP  1 2 5 0 128 255 0 1
BD  1 "Supercritical" 3 0 0 "(none)"
BCDISP  1 3 1 0 0 0 0 1
BD  1 "1D weir segment" 4 4 "Discharge coefficient" "Weir width (ft)"
"Crest level (m above sea level)" "Equation (0 = water level / 1 =
energy head)" 1 "(none)"
BV  "Discharge coefficient" 1 0 1.79769e+308
BV  "Weir width (ft)" 1 0 1.79769e+308
BV  "Crest level (m above sea level)" 0 -1.79769e+308 1.79769e+308
BV  "Equation (0 = water level / 1 = energy head)" 0 0 1
BCDISP  1 4 1 0 0 0 0 0
BD  1 "Sediment trap" 5 0 1 "Sediment transport"
```

```
BCDISP  1 5 1 0 0 0 0 1
BEDISP  2 2 0 0 0 1 1 0 0 0 0 1
BEFONT  2 1
BD  2 "2D weir" 1 3 "Discharge coefficient" "Crest level (ft above sea
level)" "Equation (0 = water level / 1 = energy head)" 1 "(none)"
BV  "Discharge coefficient" 1 0 1.79769e+308
BV  "Crest level (ft above sea level)" 0 -1.79769e+308 1.79769e+308
BV  "Equation (0 = water level / 1 = energy head)" 0 0 1
BCDISP  2 1 1 0 0 0 0 1
MD  2 "Manning" "Kinematic eddy viscosity"
MV  "Manning" 0.035 0.01 0.18
MV  "Kinematic eddy viscosity" 0 -1.79769e+308 1.79769e+308
ENDPARAMDEF
BEG2DMBC
MAT  1 0.03 20
MAT  2 0.045 20
GG "Hydrodynamic"
GP "Time interval" 20
GP "Velocity max (ft/sec)" 75
GP "H min" 0.25
GP "A min" 1
GP "Check for dry elements" 1
GP "Element style" "quadratic"
GP "Critical scour velocity" "2.0 ft/sec"
GG "Sediment transport"
GP "Time interval" 10
GP "Source X postion" 0
GP "Source Y position" 0
GP "Source elevation" 0
GP "Parcel mass (slug)" 0.5
GP "Particle mass (slug)" 0.003
GP "Particle size (in)" 0.05
GP "Deviation" 0
GP "Average density (slug/ft^3)" 3
BCN  772 3
BCN  774 3
BCN  776 3
.
.
.
BCS  4 5
BCS  1 1 80 0 1
BCE  1293 1 1 0 0
TIME 20
BCS  2 2 380
TIME 40
BCS  2 2 400
```

```
TIME 60
BCS  2 2 380
TIME 80
BCS  2 2 300
TIME 100
BCS  2 2 300
TIME 120
BCS  2 2 300
TIME 140
BCS  2 2 300
TIME 160
BCS  2 2 300
TIME 180
BCS  2 2 300
TIME 200
BCS  2 2 300
TIME 220
BCS  2 2 300
TIME 240
BCS  2 2 300
TIME 260
BCS  2 2 300
TIME 280
BCS  2 2 300
TIME 300
BCS  2 2 300
TIME 320
BCS  2 2 300
TIME 340
BCS  2 2 300
TIME 360
BCS  2 2 300
TIME 380
BCS  2 2 300
TIME 400
BCS  2 2 300
TIME 420
BCS  2 2 300
TIME 440
BCS  2 2 300
TIME 460
BCS  2 2 300
TIME 480
BCS  2 2 300
TIME 500
BCS  2 2 300
TIME 520
```

```
BCS  2 2 300
TIME 540
BCS  2 2 300
TIME 560
BCS  2 2 300
TIME 580
BCS  2 2 300
TIME 600
BCS  2 2 300
TIME 620
BCS  2 2 300
TIME 640
BCS  2 2 300
TIME 660
BCS  2 2 300
TIME 680
BCS  2 2 300
TIME 700
BCS  2 2 300
TIME 720
BCS  2 2 300
TIME 740
BCS  2 2 300
TIME 760
BCS  2 2 300
TIME 780
BCS  2 2 300
TIME 800
BCS  2 2 300
TIME 820
BCS  2 2 300
TIME 840
BCS  2 2 300
TIME 860
BCS  2 2 300
TIME 880
BCS  2 2 300
TIME 900
BCS  2 2 300
TIME 920
BCS  2 2 300
TIME 940
BCS  2 2 300
TIME 960
BCS  2 2 300
TIME 980
BCS  2 2 300
```

```
TIME 1000
BCS  2 2 300
END2DMBC
```

Related Topics

- File Formats
- Generic Model Files
- HYDRO AS-2D
- SRH-W

2D Scatter Point Files

2D Scatter Point Files (*.xy)

Two-dimensional scatter point sets are stored in 2D scatter point files. The file includes the scatter point locations and requires that functional information be defined in a separate dataset file. However, multiple scatter point sets can be stored in a single file. An XY coordinate pair defines each point in a scatter point set. The format allows time variant datasets to be associated with scattered data points as well as to organize datasets by allowing the user to assign an ID to the scattered dataset. Scatter files are opened through *File* | *Open* and are saved from *File* | *Save Scatter* from the Scatter module. When scatter point files are saved, a super file is saved. The super file saves and references the scatter and dataset files.

File Format

SCAT2D	/* File type identifier */
BEGSET	/* Beginning of cards for scatter point set */
NAME "name"	/* Name of scatter point set */
ID id	/* ID of scatter point set */
DELEV elev1	/* Default elevation */
IXY np	/* Number of points in set, begin point listing */
id1 x1 y1	/* Point id and coordinatess, one per line */
id2 x2 y2	
.	
.	
idnpxnpynp	
ENDSET	/* End of cards for scatter point set */
	/* Repeat point set cards as many times as necessary */

Sample File

```
SCAT2D
BEGSET
NAME "lakes"
ID 8493
DELEV 0.00000000000e+00
IXY 25
1 1.47000000000e+02 3.90000000000e+02
2 8.82000000000e+02 9.49000000000e+02
.
.
24 1.73000000000e+02 7.01000000000e+02
25 5.39000000000e+02 8.98000000000e+02
ENDSET
```

Cards

Card Type	**BEGSET**
Description	Identifies the beginning of a scatter point set. No fields.
Required	NO

Card Type	**NAME**		
Description	Defines the name for the following scatter point set.		
Required	NO		
Format	NAME "name"		
Sample	NAME "st mary"		
Field	*Variable*	*Value*	*Description*
1	name	str	The name for the following scatter points. Remains as default until new NAME card is encountered.

Card Type	**ID**		
Description	Defines the ID for the scatter point set.		
Required	YES		
Format	ID id		
Sample	ID 43098		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID for the following scatter point set.

Card Type	**IXY**		
Description	Defines a scatter point set.		
Required	YES		
Format	IXY np		
	id1 x1 y1		
	id2 x2 y2		
	.		
	.		
	idnp xnp ynp		
Sample	IXY 4		
	1 12.3 34.5		
	2 52.2 23.5		
	3 63.2 27.4		
	4 91.1 29.3		
Field	*Variable*	*Value*	*Description*
1	np	+	The number of scatter points in the scatter point set.
2	id	+	The ids of the points.
3-4	x,y	±	The coordinates of the points.
Repeat fields 2 - 4 np times			

Card Type	**ENDSET**
Description	Identifies the end of a scatter point set. No fields.
Required	NO

Related Topics

File Formats

ARC/INFO® ASCII Grid Files *.arc

ARC/INFO® ASCII Grid files can be opened as scatter sets in SMS. The files are opened through File | Open. To save the data, they must be saved as a scatter point file.

File Format

ncols	ncol	/* Number of columns in the grid */
nrows	nrow	/* Number of rows in the grid */
x11corner	x	/* Lower left x coordinate of grid */
y11corner	y	/* Lower left y coordinate of grid */
cellsize	size	/* Grid cell size */
NODATA_value	NODATA	/* Value of an empty grid cell */
z11 z12 z13 ... z1ncols		/* Values of row 1 */
z21 z22 z23 ... z2ncols		/* Values of row 2 */

Sample File

```
ncols          136
nrows          136
x11corner      422415
y11corner      4515405
cellsize       30
NODATA_value
1287 1286 1286 1288 ...
1288 1288 -9999 1289 ...
.
.
1282 -9999 1283 1284
```

Related Topics

- File Formats

ASCII Dataset Files *.dat

Datasets can be stored in either ASCII or binary files. Multiple datasets can be stored in a single file and both scalar and vector datasets can be saved to the same file. For scalar dataset files, one value is listed per vertex, cell, node, or scatter point. For vector dataset files, one set of XY vector components is listed per vertex, cell, node, or scatter point. If necessary, a set of status flags can be included in the file. If the status flag is false (0), the corresponding item (node, cell, etc.) is inactive. If status flags are not included in the file, it is assumed that all items are active. Dataset files are opened through File | Open and are saved when other files are saved such as 2D Scatter Point Files or through the Export Dataset Dialog.

File Format

```
DATASET                     /* File type identifier */
OBJTYPE type                /* Type of object data set is associated with */
BEGSCL                      /* Beginning of scalar data set */
OBJID id                    /* Object id */
ND numdata                  /* Number of data values */
NC numcells                 /* Number of cells or elements */
NAME "name"                 /* Data set name */
RT_JULIAN                   /* The reference time as a Julian number. */
TIMEUNITS                   /* The time units. */
TS istat time               /* Time step of the following data. */
stat1                       /* Status or activity flags */
stat2
.
.
statnumcells
val1                        /* Scalar data values */
val2
.
.
valnumdata
/* Repeat TS card for each time step */
ENDDS                       /* End of data set */
BEGVEC                      /* Beginning of vector dataset */
VECTYPE type                /* Vector at node/gridnode or element/cell */
OBJID id                    /* Object id */
ND numdata                  /* Number of data values */
NC numcells                 /* Number of cells or elements */
NAME "name"                 /* Data set name */
TS istat time               /* Time step of the following data. */
stat1                       /* Status or activity flags */
stat2
.
.
statnumcells
vx1 vy1
```

```
vx2 vy2
.
.
vnumdata vnumdata vnumdata
/* Repeat TS card for each time step */
ENDDS                 /* End of data set */
/* Repeat BEGSCL and BEGVEC sequences for each data set */
```

Sample File

Note: This sample file is using an activity array, so there are 16 values per TS. The first 8 values are activity flags for each of the 8 nodes. Values 9-16 are the scalar values for the 8 nodes.

```
DATASET
OBJTYPE "grid2d"
BEGSCL
ND 8
NC 8
NAME "sediment transport"
RT_JULIAN 2453867.068720
TIMEUNITS seconds
TS 1 1.00000000e+00
0
0
0
1
1
1
1
0
0.00000000e+00
0.00000000e+00
0.00000000e+00
3.24000000e+00
4.39000000e+00
2.96000000e+00
7.48000000e+00
0.00000000e+00
ENDDS
BEGVEC
VECTYPE 0
ND 8
NC 8
NAME "velocity"
TS 1 5.00000000e+00
0
0
0
```

```
1
1
1
1
0
1.60000000e+01 1.60000000e+01
6.40000000e+01 6.40000000e+01
1.44000000e+02 1.44000000e+02
1.96000000e+02 1.96000000e+02
2.25000000e+02 2.25000000e+02
9.21600000e+03 9.21600000e+03
9.60400000e+03 9.60400000e+03
9.80100000e+03 9.80100000e+03
ENDDS
```

Cards

Card Type	**DATASET**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**OBJTYPE**		
Description	Identifies the type of objects that the datasets in the file are associated with.		
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.		
Format	OBJTYPE type		
Sample	OBJTYPE tin		
Field	*Variable*	*Value*	*Description*
1	type	"mesh2d" "scat2d" "cgrid2d" "specgrid2d"	2D mesh 2D scatterpoints 2D cartesian grid Spectral energy grid

Card Type	**OBJID**		
Description	Identifies the object that the data sets in the file are associated with.		
Required	NO. Card is only used if the OBJTYPE is scat2d.		
Format	OBJID id		
Sample	OBJID 1254		
Field	*Variable*	*Value*	*Description*
1	id	Integer > 0	The ID of the object.

Card Type	**RT_JULIAN**			
Description	The reference time as a Julian number.			
Required	NO			
Field	*Variable*	*Size*	*Value*	*Description*
1	reference time	8 byte float	+/-	Continuous count of days and fractions of days since noon Universal Time on January 1, 4713 BCE (on the Julian calendar).

Card Type	**TIMEUNITS**			
Description	The time units.			
Required	NO, but recommended			
Field	*Variable*	*Size*	*Value*	*Description*
1	time units	4 byte float	0	Hours
			1	Minutes
			2	Seconds

Card Type	**BEGSCL**
Description	Scalar data set file identifier. Marks beginning of scalar data set. No fields.
Required	YES

Card Type	**BEGVEC**
Description	Vector data set file identifier. Marks beginning of vector data set. No fields.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors are applied to the nodes/grid nodes. The vectors are applied to the elements/cells.

Card Type	**ND**		
Description	The number of data values that will be listed per time step. This number should correspond to the total number of vertices, nodes, cells centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.		
Required	YES.		
Format	ND numdata		
Sample	ND 10098		
Field	*Variable*	*Value*	*Description*
1	numdata	+	The number of items. At each time step, numdata values are printed.

Card Type	**NC**		
Description	This number should correspond to the maximum element id (meshes) or the number of cells (grids).		
Required	YES.		
Format	NC numcells		
Sample	NC 3982		
Field	*Variable*	*Value*	*Description*
1	numcells	+	The number of elements or cells

Card Type	**NAME**		
Description	The name of the data set.		
Required	YES.		
Format	NAME "name"		
Sample	NAME "Total head"		
Field	*Variable*	*Value*	*Description*
1	"name"	str	The name of the dataset in double quotes.

Card Type	**TS**		
Description	Marks the beginning of a new time step, indicates if stat flags are given, and defines the time step value, status flags, and scalar data values for each item.		
Required	YES.		
Format	TS istat time stat1 stat2 . . stat numcells val1 val2 . . valnumdata		
Field	*Variable*	*Value*	*Description*
1	active array flag	0 1	Activity or status values not specified Activity or status values specified - A value of 0 (inactive) or 1 (active) must be specified for each node following the TS card, before the scalar or vector values are specified.
2	Time step	double	Time step value for data

Related Topics

- File Formats
- Binary Dataset Files (*.dat)
- Export Dataset Dialog

Binary Dataset Files *.dat

Datasets can be stored to either ASCII or binary files. Compared to ASCII files, binary files require less memory and can be imported to SMS more quickly. The disadvantages of binary files are that they are not as portable and they cannot be viewed with a text editor. The binary format is patterned after the ASCII format in that the data are grouped into "cards". However, the cards are identified by a number rather than a card title.

Dataset files are opened through File | Open and are saved when other files are saved such as 2D Scatter Point Files.

File Format

Card	Item	Size	Description
	version	4 byte integer	The SMS binary dataset file format version. value = 3000
100	objecttype	4 byte integer	Indentifies the type of objects that the datasets in the file are associated with. Options are as follows: 1 TINs 3 2D meshes 5 2D scatter points
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).
120	SFLG	4 byte integer	The number of bytes that will be used in the remainder of the file for status flags.
110	SFLT	4 byte integer	The number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).
130 or 140	BEGSCL or BEGVEC		The number of bytes that will be used in the remainder of the file for status flags.
150	VECTYPE	4 byte integer	(0 or 1) In the case of vector dataset files, indicates whether the vectors will be applied at the nodes/gridnodes or the elements/cells.
160	OBJID	4 byte integer	The id of the associated object. Value is ignored for grids and meshes.
170	NUMDATA	4 byte integer	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid) or scatter points.
180	NUMCELLS	4 byte integer	This number should correspond to the number of elements (meshes) or the number of cells (mesh-centered grids). Value is ignored for other object types.
190	NAME	40 bytes	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.
200	TS		Marks the beginning of a time step.
	ISTAT	SFLG integer	(0 or 1) Indicates whether or not status flags will be included in the file.
	TIME	SFLT real	Time corresponding to the time step.
	statflag1	SFLG integer	Status flag (0 or 1) for node 1
	statflag2	SFLG integer	Status flag (0 or 1) for node 2
	...		

Repeat card 200 for each time step in the dataset.			
210	ENDDS		Signal the end of a set of cards defining a dataset.
240	RT_JULIAN	8 byte float	The reference time as a Julian number.
250	TIMEUNITS	4 byte integer	The time units as follows: 0 - hours 1 - minutes 2 - seconds 4 - days

Cards

Card Type	**VERSION**
Card ID	3000
Description	File type identifier. No fields.
Required	YES

Card Type	**OBJTYPE**			
Card ID	100			
Description	Identifies the type of objects that the datasets in the file are associated with.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The data sets would then be assigned to the objects corresponding to the active module.			
Format	OBJTYPE type			
Sample	OBJTYPE tin			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	tin mesh2d scat2d	Tins 2D mesh 2D scatterpoints

Card Type	**SFLT**			
Card ID	110			
Description	Identifies the number of bytes that will be used in the remainder of the file for each floating point value (4, 8, or 16).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizefloat	4 byte int	4 8 or 16	Number of bytes

Card Type	**SFLG**			
Card ID	120			
Description	Identifies the number of bytes that will be used in the remainder of the file for status flags (1, 2, or 4).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	sizeflag	4 byte int 1 2 or 4	Number of bytes	

Card Type	**BEGSCL**
Card ID	130
Description	Marks the beginning of a set of cards defining a scalar dataset.
Required	YES

Card Type	**BEGVEC**
Card ID	140
Description	Marks the beginning of a set of cards defining a vector dataset.
Required	YES

Card Type	**VECTYPE**			
Card ID	150			
Description	Identifies the type of vector data that will be read and where to apply it.			
Required	YES. If card does not exist, the file can only be read through the Data Browser. The datasets would then be assigned to the objects corresponding to the active module.			
Field	*Variable*	*Size*	*Value*	*Description*
1	type	4 byte int	0 1	The vectors are applied to the nodes/grid nodes. The vectors are applied to the elements/cells.

Card Type	**OBJID**			
Card ID	160			
Description	The id of the associated object.			
Required	This card is required in the case of TINs, 2D scatter points, and 3D scatter points. With each of these objects, multiple objects may be defined at once. Hence the id is necessary to relate the dataset to the proper object.			
Field	*Variable*	*Size*	*Value*	*Description*
1	id	4 byte int	+	The id of the object.

Card Type	**NUMDATA**			
Card ID	170			
Description	The id of the associated object.			
Required	The number of data values that will be listed per time step. This number should correspond to the number of vertices, nodes, cell centers (cell-centered grid), cell corners (mesh-centered grid), maximum node id (meshes) or scatter points.			
Field	*Variable*	*Size*	*Value*	*Description*
1	numdata	4 byte int	+	The number of items. At each time step, numdata are listed.

Card Type	**NUMCELLS**			
Card ID	180			
Description	This number should correspond to the element id (meshes) or the number of cells (grids).			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	numcells	4 byte int	+	The number of elements or cells.

Card Type	**NAME**			
Card ID	190			
Description	The name of the data set.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	name	40 bytes	str	The name of the dataset. Use one character per byte. Mark the end of the string with the '\0' character.

Card Type	**TS**			
Card ID	200			
Description	Defines the set of scalar values associated with a time step. Should be repeated for each time step.			
Required	YES			
Field	*Variable*	*Size*	*Value*	*Description*
1	istat	SFLG int	0 1	Use status flags from previous time step. For the first time step, this value indicates that all cells are active. Status flags will be listed.
2	time	SFLT int	+	The time step value. This number is ignored if there is only one time step

	stat	SFLG int	0 1	Inactive Active One status flag should be listed for each cell or element. These flags are included only when istat = 1.
	val	SFLT real	+/-	The scalar values.

Card Type	**ENDDS**
Card ID	210
Description	Signals the end of a set of cards defining a dataset
Required	YES

Card Type	**RT_JULIAN**			
Card ID	240			
Description	The reference time as a Julian number.			
Required	NO			
Field	*Variable*	*Size*	*Value*	*Description*
1	reference time	8 byte float	+/-	Continuous count of days and fractions of days since noon Universal Time on January 1, 4713 BCE (on the Julian calendar).

Card Type	**TIMEUNITS**			
Card ID	250			
Description	The time units.			
Required	NO, but recommended			
Field	*Variable*	*Size*	*Value*	*Description*
1	time units	4 byte float	0 1 2	Hours Minutes Seconds

Related Topics

- File Formats
- ASCII Dataset Files (*.dat)

Boundary ID Files

Boundary ID Files (*.bid)

Boundary ID files are used to store the ids of the nodes on all of the nodestrings in a mesh. The idea of this file is to export only the boundary ids. To do this, make sure nodestrings only exist along the boundary. Boundary ID files are not opened in SMS but are saved using the menu command File | Save As... in the Mesh module. To save a Boundary ID File (*.bid), the current mesh module model should be set to ADCIRC. Use the menu command "Data |Switch Current Model..." to change the current mesh module model.

Sample File

```
BOUNDARY
3 Number of nodestrings
3 Number of nodes in nodestring
24
18
12
5 Number of nodes in nodestring
149
49
43
37
31
8 Number of nodes in nodestring
11
112
120
127
134
141
148
155
```

Related Topics

- Boundary XY Files
- File Formats

Boundary XY Files

Boundary XY Files (*.bxy)

Boundary XY files are used to store the x,y location of the nodes on all the nodestrings in a mesh. If you would like to export only the boundary node locations, make sure nodestrings only exist on the boundary of the mesh. If nodestrings exist on the interior of the mesh, their locations will also be included in the Boundary XY file. Boundary ID files are not opened in SMS but are saved using the menu command File | Save As... in the Mesh module. To save a Boundary XY File (*.bxy), the current mesh module model should be set to ADCIRC. Use the menu command "Data | Switch Current Model..." to change the current mesh module model.

Sample File

```
BOUNDARY
2 Number of nodestrings
3 Number of nodes in nodestring
500000.000000 0.000000
497592.400000 49008.600000
490392.600000 97545.200000
5 Number of nodes in nodestring
518643.200000 65498.400000
654128.400000 54688.400000
845478.800000 53491.100000
987681.700000 65385.800000
551972.600000 48687.200000
```

Related Topics

- Boundary ID Files
- File Formats

Coastline Files *.cst

Coastline files contain x, y, z location information for arcs defining coastlines. The arcs defining a coastline can be created from ADCIRC and CGWAVE type coverages. When a Coastline file is read into SMS, feature arcs are created. If a coastline closes, the final point is not repeated. If no z-value is specified, SMS defaults the node z-value to 0.0.

Coastline files are opened through the menu File | Open and saved from File | Save As... from the Map module.

Sample File

COAST	/* File type identifier */
2	/* Number of coastlines */
1309 0	/* Number of segments in coastline and if coastline closes (not closes = 0, closes = 1) */
-7794.9054 3396.0346 0.0	/* Node X, Y, Z Locations - Z is optional (defaults to 0.0) */
-7822.6129 3391.8341 0.0	
-7852.6508 3386.68 0.0	
.	
.	
.	
151 1	/* Start next coastline */
.	
.	
.	
/* EOF */	

Related Topics

- Create Coastline
- File Formats
- ADCIRC
- CGWAVE

Color Palette Files *.pal

Color Palette files hold user defined palette information. The process for creating a color palette is described in the User Defined Color Palettes article.

A sample palette file is shown below:

```
PALETTE                    File type identifier
2                          Number of Palettes
Palette1  3                Name of Palette; Number of colors in palette
0 0.0  1.0                 Percentage (0) or Value (1); Min Range; Max Range
0.111111 0 0  0            1st color - Percentage; Red (0-255); Green; Blue
0.444444 45  136 45        2nd color
1.000000 255  255 255      3rd color
Palette2  5                Next Palette
...                        etc.
```

Related Topics

- File Formats
- User Defined Palettes
- Color Options Dialog
- New Palette Dialog

Drogue Files *.pth

Drogue files contain particle/path data. Drogue plots are generated by the ADCIRC model.

Drogue files are opened through File | Open but are not saved from SMS.

Sample File

```
ACE/vis drogue path file   /* Title */
5                          /* Number of Time Steps */

7200.0000   199   /* Current Time Step and Number of Particles */

-0.766993322E+02   0.346589454E+02   1   /* xy values and id */
-0.766986001E+02   0.346616775E+02   2

.
.

8000.0000   199   /* Next Time Step and Number of Particles */
```

.

.

/* EOF */

The number of particles must be the same for each time step.

Related Topics

- File Formats

File Extensions

Model specific files such as those used by FESWMS, RMA2, and other models are documented in their respective reference documentation, which is available from the model developers, not the developers of the SMS software.

OPEN/SAVE FILES

- ASCII Data (*.dat)
- AutoCAD (*.dxf)
- Binary Data (*.dat)
- Bitmap Image Files (*.bmp)
- Coastline (*.cst)
- Image (*.img)
- INI / Settings (*.ini)
- JPEG Image Files (*.jpg, *.jpeg)
- Map (*.map)
- Material (*.mat)
- Meta Data (*.met)
- Observation Table (*.obt)
- Palette File (*.pal)
- Project (*.spr; *.prj)
- Scatter Point (*.xy)
- TIN (*.tin)
- 2D Mesh (*.2dm)
- XY Series (*.xys)
- XYZ (*.xyz)

OPEN ONLY

- Arc Info (*.arc)
- Shape File (*.shp)

ADCIRC

- ADCIRC Control (*.ctl; *.15)
- ADCIRC Harmonic Solution (*.53; *.54)
- ADCIRC Simulation (*.grd; *.14)
- ADCIRC Unit 63 (*.63; *.sol)
- ADCIRC Unit 64 (*.64; *.sol)
- ADCIRC Unit 71 (*.71; *.sol)
- ADCIRC Unit 72 (*.72; *.sol)
- ADCIRC Unit 73 (*.73; *.sol)
- ADCIRC Unit 74 (*.74; *.sol)
- ADCIRC Unit 75 (*.75; *.sol)
- ADCIRC Unit 83 (*.83; *.sol)
- ADCIRC Unit 84 (*.84; *.sol)
- ADCIRC Unit 85 (*.85, *.sol)
- ADCIRC Unit 91 (*.91, *.sol)
- ADCIRC Unit 93 (*.93, *.sol)
- Drogue (*.pth)

CGWAVE

- CGWAVE Simulation (*.cgi)
- CGWAVE Solution (*.cgo; *.out)
- CGWAVE 1-D (*.cg1)

CH3D

- CH3D Simulation (*.ch3)

CMS-Wave

- CMS-Wave Simulation (*.sim)
- CMS-Wave Current File (*.cur)
- CMS-Wave Depth File (*.dep)

DAMBRK

- DAMBRK Simulation (*.dat)

FESWMS

- FESWMS Simulation (*.fil)
- FESWMS Solution (*.flo; *.out)

GENESIS

- GENESIS Solution (*.sol)

STWAVE

- STWAVE Current (*.cur)
- STWAVE Depth (*.dep)
- STWAVE Observation (*.obs)
- STWAVE Simulation (*.sim)
- STWAVE Spectral Energy (*.eng)
- STWAVE Wavefield (*.wav)

TABS

- RMA2 BC (*.bc)
- RMA2 Geometry (*.geo)
- RMA2 Simulation (*.sim)
- RMA2 Solution (*.sol)
- RMA4 Simulation (*.trn)
- RMA10 Simulation (*.bc)

WSPRO

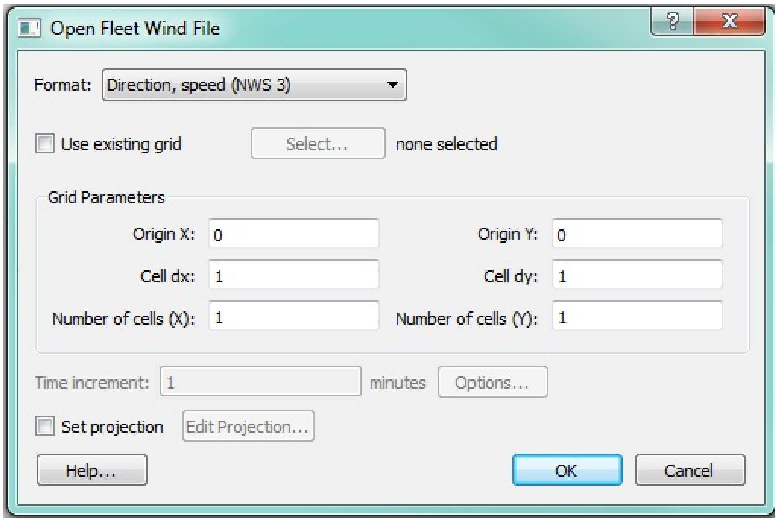
- WSPRO Simulation (*.dat)
- WSPRO Solution (*.out)

Related Topics

- File Formats

Fleet Wind Files

Fleet Wind Files (*.fleetwind)



Fleet Wind Dialog

Datasets can be read in from fleet wind files. SMS supports two different formats for fleet wind files, which are equivalent to NWS = 3 and NWS = 6 for PBL models. More information regarding the format for these files can be found at http://adcirc.org/documentv49/fort_22.html [1].

Opening the Files

When opening a fleet wind file, SMS will bring up a dialog to prompt the user for information regarding file type, grid parameters, timesteps, and projection.

Format: Option to select between "Direction, speed (NWS 3)" and "X, Y, pressure (NWS 6)"

Existing Grid: If the grid specified in the fleet wind file already exists in SMS, you can select to create the datasets on the existing grid

Grid Parameters: If the grid specified in the fleet wind file does not already exist, you must specify the origin, cell size, and number of cells to create the grid.

Time (NWS 6 only): Specify the size and units of the time increment, as well as a reference time (if desired).

Projection: Specify the projection of the grid. If the projection is not specified, the grid will be created using the current projection.

Datasets

When SMS reads a fleetwind file, it will create the following datasets on the grid:

- Wind (vector)
- Magnitude (scalar)
- Pressure (scalar) – NWS 6 only

References

[1] http://adcirc.org/documentv49/fort_22.html

Importing Non-Native SMS Files

SMS can import many files generated by other software in their native format. The files that can be imported to SMS are shown in the tables below. Each file type is identified by the file extension. The file filter corresponding to the desired extension should be selected in the Open File dialog. In addition to the file types listed below, several other types of data can be imported via the Import Wizard. Refer to the Import Wizard article for more information.

Import File Types

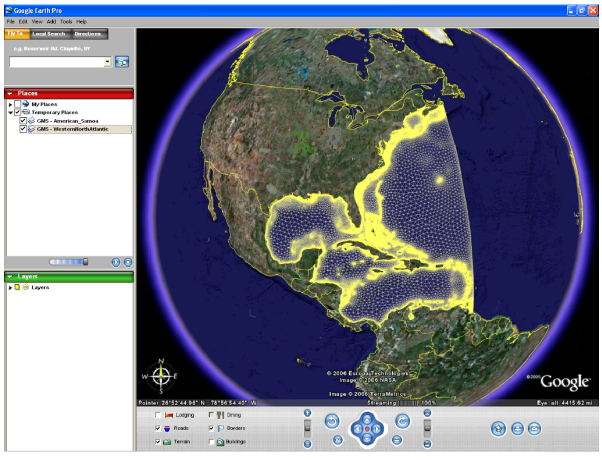
File Type	File Extension	Purpose
ARC/INFO® ASCII Grid Files	*.arc	Cartesian grid and associated data sets
Drogue Files	*.pth	Particle / path data from ADCIRC

Related Topics

- File Import Wizard
- Native File Formats
- File Formats

KMZ Files

XMS software can export KMZ files. KMZ [1] files can be imported into Google Earth [2]. KMZ files cannot currently be imported into XMS.



A 2D Mesh inside Google Earth

Raster vs. Vector

The KMZ file format supports both vector data (lines, points, polygons, triangles etc.) and raster data (images). When exporting raster data, the image shown in the XMS main graphics window is saved as a PNG image file with georeferencing data. The data is clipped to match the window bounds of the current view. When exporting vector data, all data displayed, as specified by the display options and project explorer, is exported to a raster KMZ file. The following versions of XMS software support vector export:

- GMS – 7.1
- SMS – 10.1
- WMS – 8.1

How To Export - Screen Capture

- You must be in plan view before you can export a KMZ file.
- You can export a KMZ file by using the standard *File|Save As* dialog and selecting either the **Google Earth© Raster KMZ File (*.kmz)** or **Google Earth© Vector KMZ File (*.kmz)** option in the **Save as type** field.
- To specify a resolution higher than the screen resolution:
  - GMS – Select the **Options** button in the *Save As* dialog.
  - SMS – Change the copy to clipboard scale factor in the Preferences dialog

The background color is made transparent in the exported KMZ file so the Earth can be seen through the overlayed image in Google Earth©.

How To Export – Transient Data Animation

Export a KMZ transient data animation using the Film Loop Wizard. This will export a series of raster images which can be animated in Google Earth©. The background color can be specified in the Film Loop Wizard. It is recommended that the option for no background be used so the Earth can be seen through the overlayed image in Google Earth©. You must be in Plan View to export a KMZ transient data animation.

By default, Coordinated Universal Time (UTC) [2] is assumed when exporting KMZ files. An offset from UTC can be specified. A list of time zone offsets from UTC is given here [3].

See "Viewing a Timeline" on page 90 of the Google Earth© user Guide [4] or here [5] for an explanation of how to change the time zone used by Google Earth©.

Coordinate System

KMZ files contain latitude and longitude information to define the location of the image. If your current coordinate system type is a projection, like UTM for example, and not a geographic system (which uses latitude and longitude), XMS will attempt to determine the latitude and longitude using coordinate conversion. It's possible that the coordinate conversion may fail, or that the distortion from converting from the current coordinate system to a geographic system is too high. In either case, XMS will issue a warning.

Transparency

The background color is made transparent in the exported PNG image which is in the KMZ file. This makes it so that the Earth can be seen through the overlayed image.

See Also

- Official Google Earth website [2]
- KML documentation [1]

Back to XMS

References

[1] http://code.google.com/apis/kml/documentation/
[2] http://en.wikipedia.org/wiki/Coordinated_Universal_Time
[3] http://en.wikipedia.org/wiki/Timezones
[4] http://earth.google.com/userguide/v4/google_earth_user_guide.pdf
[5] http://earth.google.com/userguide/v4/ug_gps.html

LandXML Files

LandXML Files (*.xml)

A LandXML file is a non-proprietary file format that stores civil/survey data such as points, faces, etc. making it easier to share surfaces between different programs. Several CAD and other packages support exporting data into the LandXML format. This makes LandXML a good choice for getting bathymetry/topographic data into SMS as the connectivity will be maintained.

SMS supports importing TINs from a LandXML file.

Locating Data in a LandXML file

SMS only supports a subset of the LandXML definition specific to TINs. The specific data SMS looks for point location, and triangular connectivity information. To locate this data within the LandXML file, start by looking under the identifier "Definition surfType". Make sure that this is defined as "TIN". Then, look for the 'Pnts' identifier to read point locations by id. Next, look for the 'Faces' identifier to read the connectivity information. Note: In the LandXML file, the points are identified by their id number as well as by their coordinates (y,x,z) respectively.

LandXML Identifiers

A complete list of LandXML identifiers with their respective definitions can be found at www.landxml.org/ [1] on the right side of the page under LandXML-1.2 Schema. Click on the LandXML-1.2 Data Diagram.

References

[1] http://www.landxml.org/

Map Files

Related Topics

- File Formats

Material Files *.mat

Each element of a 2D mesh has an assigned material ID. Specific material properties are related to the analysis models, and are stored in the analysis files. However, general material properties, such as color, are not stored in these files. Therefore, they are stored in the material file. A material ID represents an index to a global list of materials. The material file associates general attributes such as a name, color, and pattern with each of the materials.

Material files are opened through File | Open and are saved from File | Save As... from the Mesh module.

File Format

MAT	/* File type identifier */
MN id name	/*Material name */
MC id red green blue	/* Material color */
MS id stippleid	/* Material stipple (fill pattern) */

Cards

Card Type	**MAT**
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**MN**		
Description	Identifies a name to be associated with the material.		
Required	NO		
Format	MN id name		
Sample	MN 5 bedrock		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID of the material.
2	name	str	The name of the material.

Card Type	**MC**		
Description	Identifies a color to be associated with the material.		
Required	NO		
Format	MN id red green blue		
Sample	MN 5 124 67 245		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID of the material.
2	red	0-255	The value of the red component of the color.
3	green	0-255	The value of the green component of the color.
4	blue	0-255	The value of the blue component of the color.

Card Type	**MS**		
Description	Identifies a stipple (fill pattern) to be associated with the material. This stipple is used whenever an object is being drawn using color filled polygons.		
Required	NO		
Format	MN id stippleid		
Sample	MN 5 13		
Field	*Variable*	*Value*	*Description*
1	id	+	The ID of the material.
2	stippleid	+	The ID of the stipple.

Related Topics

- File Formats

Native SMS Files

Native SMS Files are files SMS can read and write.

SMS Interface Files

SMS interface files contain data used by SMS to save display options, user preferences, etc.

SMS Interface Files include:

File Type	File Extension	Purpose
Color Palette Files	*.pal	User defined contour palette information
Settings (INI) Files	*.ini	SMS preferences and default settings

Generic Data Files

A generic file is defined as any file that was not formatted for a specific numeric model. Model specific files such as those used by FESWMS, RMA2, and other models are documented in their respective reference documentation, which is available from the model developers, not the developers of the SMS software.

Most of the generic files used by SMS use a modified form of the HEC style card type format. With this format, the different components of the file are grouped into logical groups called "cards." The first component of each card is a short name that serves as the card identifier. The remaining fields on the line contain the information associated with the card. In some cases, such as lists, a card can use multiple lines.

While card style input makes the file slightly more verbose, there are many advantages associated with the card type approach to formatting files. Some of the advantages are:

- Card identifiers make the file easier to read. Each input line has a label, which helps to identify the data on the line.
- The card names are useful as text strings for searching in a large file. All input lines of a particular type can be located quickly in a large input file.
- In many cases, Cards allow the data to be input in any order (i.e., the order that the cards appear in the file is usually not important).
- Cards make it easy to modify a file format. New card types can be added without invalidating older files. New files have additional data in the new cards. The new card must be optional (which is typically the case for new cards) for old files to remain compatible. If an old card type is no longer used, the card can simply be ignored without causing input errors.

The generic file formats supported by SMS include:

File Type	File Extension	Purpose
2D Mesh Files	*.2dm	Finite element mesh and generic model interface template definition
2D Scatter Point Files	*.xy	Scatter point x, y location
ASCII Dataset Files	*.dat	Mesh, grid, or scatter set datasets
Binary Dataset Files	*.dat	Mesh, grid, or scatter set datasets
Boundary ID Files	.bid	Export node ids of nodestring nodes
Boundary XY Files	.bxy	Export node x, y locations of nodestring nodes
Coastline Files	*.cst	Export node and vertice x, y locations of coastline arcs
Map Files	*.map	File that contains information for feature data
Material Files	*.mat, *.materials	Mesh material options for SMS (display options, name, etc.)
Quad4 Files	*.qd4	QUAD-4 format finite element mesh file
Shapefiles	*.shp	GIS vector data file
SMS Super Files	*.sup	File containing references to geometry and dataset files
Tabular Data Files – SHOALS	*.pts	Mesh node, or scatter point locations and values
TIN Files	*.tin	Triangulated irregular network data
XY Series Files	*.xys	X, Y series data curves
XYZ Files	*.xyz	Mesh node, or scatter point locations and values

Related Topics

- Import Wizard
- Non-native SMS Files
- File Formats

Quad4 Files

QUAD4 files are exported from SMS for use with a proprietary finite element program called QUAD-4. The file format is a specific format listed in the QUAD-4 documentation.

Related Topics

- File Formats

External Links

- Idriss, I.M., Lysmer, John, Hwang, Richard N., and Seed, H. Bolton, "QUAD-4, A Computer Program for Evaluating the Seismic Response of Soils Structures by Variable Damping Finite Elements, "Earthquake Engineering Research Center, Report No. EERC 73-16, University of California, Berkeley, June 1973.
- The Earthquake Engineering Online Archive QUAD-4 and QUAD4M Software and Manuals [1]

References

[1] http://nisee.berkeley.edu/elibrary/getpkg?id=QUAD4

Settings Files *.ini

Settings, or INI, files store all of the settings that the user sets inside of SMS. For example, the user defined display options, the coordinate system, mesh, grid, and scatter options, etc. are stored in the file. These files should not be confused with the old .ini files that were previously used as FESWMS Initial Condition files.

SMS.INI

The main settings file, called sms.ini, is created when SMS is started for the first time. The file is then opened every time SMS starts. When the user makes a change, such as setting mesh nodes to draw in blue, the change is permanently saved by invoking the File | Save Settings command, which updates the sms7.ini file. If the user deletes the sms7.ini, the next time that SMS begins, the settings will be reset to the factory defaults.

Other Settings Files

Settings files are also saved with a project file. The next time a project is loaded into SMS, the project settings are restored and the SMS environment returns to its saved state.

File Format

Settings files are ASCII text files. Each SMS setting is stored with a keyword and a value. For example:

```
Show Welcome Dialog=0
```

signifies that SMS will not show the welcome dialog on startup.

Related Topics

- File Formats

Shapefiles

One common method for creating feature objects is to import a shapefile. The concept of a shapefile was established by Environmental Systems Research Institute (ESRI) [1] in their ArcView® program and it has become the defacto standard for sharing GIS vector data (points, lines, and polygons).

Shapefiles contain data exported from ARC/INFO® or ArcView® in binary format. When they are imported into SMS, the data is converted to feature objects, points, arcs, or polygons. Shapefiles are opened through File | Open. Only map data can be saved out from SMS as a shapefile.

A shapefile is actually comprised of three or more files. The primary file is the *.shp and it contains the geometric information (coordinates and if necessary connectivity of the points, lines, polygons). The *.dbf file is a standard database file and stores the attributes of the feature objects. Finally there will be a *.shx file which is an indexing file. There may be a few other files that accompany the shapefile and so you should always move them around together if you are copying or moving them to a new directory.

Only one "theme" or type of feature can exist in a shapefile. For example you cannot store points and polygons in the shapefile, or streams and basin boundaries and so you may be required to import multiple files to make up the drainage coverage in SMS.

When a shapefile is opened, the Import Shapefile Data dialog appears. The user must know if the file is a Point, Arc, or Polygon shapefile. The options in the dialog are:

- **Coverage Options** – Bring up the coverage options dialog to set the coverage in which the shapefile data is created.
- **Points / Arcs / Polygons** – Under each of these sections, the file browser button allows the user to select a point, arc, and polygon file. One file of each type may be selected.
- **Attribute Mapping** – This button brings up the Map Shapefile Attributes dialog. In this dialog, the user can select an attribute from each of the Database (from the shapefile) and Coverage attributes (SMS supported attributes) fields. If an attribute is selected from each, the Map button maps the attributes and the Unmap button unmaps the attributes. If an attribute is mapped, the attributes will be assigned in SMS when the file is opened.

SMS includes all of the tools necessary to import shapefiles and convert the geometric and attribute information into feature objects. This can be done by directly opening the shapefile and converting to feature objects in the active coverage or by loading the shapefile in the GIS module.

Related Topics

- Data Acquisition
- Importing Shapefiles
- GIS Module
- File Formats
- Feature Objects

SMS Super Files *.sup

SMS Super files were used in previous versions of SMS to save most of the working data in SMS. Super files have been replaced by Project files. Old super files are still opened in SMS. Super files are only saved from the File | Save Scatter Super File command. These contain a 2D Scatter Point file and the corresponding ASCII data file.

A super file contains a list of other files. Each of the files in the list must be one of the basic SMS file types (2D meshes, 2D scatter points, materials, TINs). If a super file is selected using the File | Open command, each of the files listed in the super file are opened and imported. This makes it possible to quickly read in several files without having to identify each file individually in the file browser.

The file format for a super file is shown below. The first line in the file is the SUPER card, which identifies the file as a super file. Each of the other cards shown are optional. Each of the file cards has a card identifier representing the type of file. The identifier is followed by a file name. The file name should be a complete path if the file is not in the same directory as the super file. Any suffix may be used for the file name.

File Format

SUPER	/* File type identifier */
MAT filename	/* Material File */
SCAT2D filename	/* 2D scatter point file */
MAP filename	/* Map file */
MESH2D filename	/* 2D Mesh file */
DATA filename	/* Dataset File */
STNGS filename	/* Settings (*.ini) File */
IMAGE filename	/* Image file */

Sample File

```
SUPER
MAT    c:\SMS\DATA\SITE1\site1.mat
SCAT2D c:\SMS\DATA\SITE1\site1.xyf
```

Related Topics

- File Formats

Tabular Data Files - SHOALS *.pts

SMS includes the ability to import and export tabular data files. These files can include any number of columns of data. The user selects which columns are to be imported and how they are to be interpreted. The columns of data follow an optional header and may be delimited by any character (such as TAB, SPACE, COMMA, etc.). Files with the extensions of *.xyz and *.pts are defaulted to be of this type.

History

This capability was originally developed to support files generated by the SHOALS group of the US Army Corps of Engineers [1]. SHOALS files generally have an optional file header describing the data in each column of the file.

Importing Tabular Data

Tabular data files are opened using the File | Open menu command. The user selects the tabular data file to open. The Text Import Wizard appears and allows the user to select how the data should be interpreted (as scatter points, mesh nodes, or map nodes) and which columns should be imported.

Exporting Tabular Data

Tabular data files are saved using the File | Save As... menu command and selecting *Tabular Data Files (*.txt)* or *Shoal Files (*.pts)* from the save as type filter combo box.

When in the Scatter module this saves scattered data vertices to the tabular file. When in the Mesh module this save the mesh nodes into the tabular data file. In either case the Export Tabular File dialog appears to support this operation.

Related Topics

- File Formats

TIN Files

TIN files are used for storing triangulated irregular networks. The TIN file format is shown below and a sample file is shown after. The TIN file format can be used to import a simple set of xyz coordinates since the triangle information (beginning with the TRI card) does not need to be present. If you have a file of xyz coordinates you only need to add the TIN, BEGT, and VERT nv cards to the top of the file and the ENDT card at the end.

```
TIN                  /* File type identifier */
BEGT                 /* Beginning of TIN group */
TNAM name            /* Name of TIN */
TCOL id              /* TIN material id */
VERT nv              /* Beg. of vertices */
x1 y1 z1 lf1         /* Vertex coords. */
x2 y2 z2 lf2
.
.
.
xnv ynv znv lfnv
TRI nt               /* Beg. of triangles */
v11 v12 v13          /* Triangle vertices */
v21 v22 v23
.
.
.
vnt1 vnt2 vnt3
ENDT                 /* End of TIN group */
```

Sample TIN File:

```
TIN
BEGT
TNAM Aspen
TCOL 255 255 255
VERT 408
0.0 3.1 7.8 0
5.3 8.7 4.0 1
.
.
2.4 4.4 9.0 1
TRI 408
5 1 4
4 1 2
.
.
4 2 3
ENDT
```

Cards used in the TIN file

Card Type	**TIN**
Card ID	3000
Description	File type identifier. Must be on first line of file. No fields.
Required	YES

Card Type	**BEGT**
Card ID	3000
Description	Marks the beginning of a group of cards describing a TIN. There should be a corresponding ENDT card at a latter point in the file. No fields.
Required	YES

Card Type	**TNAM**		
Description	Provides a name to be associated with the TIN.		
Required	NO		
Format	TNAM name		
Sample	`TNAM aspen`		
Field	**Variable**	**Value**	**Description**
1	name	str	The name of the TIN.

Card Type	**TCOL**		
Description	Defines a default color for the triangles of the TIN		
Required	NO		
Format	TCOL color_red color_green color_blue		
Sample	`TCOL 255 255 255`		
Field	**Variable**	**Value**	**Description**
1	color_red	0–255	The red color component of TIN triangles.
2	color_green	0–255	The green color component of TIN triangles.
3	color_blue	0–255	The blue color component of TIN triangles.

Card Type	**MAT**		
Description	Associates a material id with the TIN. This is typically the id of the material which is below the TIN.		
Required	NO		
Format	MAT id		
Sample	MAT 3		
Field	**Variable**	**Value**	**Description**
1	id	+	The material ID.

<table>
<tr><th>Card Type</th><th colspan="3">VERT</th></tr>
<tr><th>Description</th><td colspan="3">Lists the vertices in the TIN</td></tr>
<tr><th>Required</th><td colspan="3">YES</td></tr>
<tr><th>Format</th><td colspan="3">VERT nv
x_1 y_1 z_1 lf_1
x_2 y_2 z_2 lf_2
.
.
x_{nv} y_{nv} z_{nv} lf_{nv}</td></tr>
<tr><th>Sample</th><td colspan="3"><code>VERT 4</code>
<code>0.0 3.1 7.8 0</code>
<code>5.3 8.7 4.0 1</code>
<code>2.4 4.4 9.0 1</code>
<code>3.9 1.2 3.6 0</code></td></tr>
<tr><th>Field</th><th>Variable</th><th>Value</th><th>Description</th></tr>
<tr><td>1</td><td>nv</td><td>+</td><td>The number of vertices in the TIN</td></tr>
<tr><td>2–4</td><td>x,y,z</td><td>±</td><td>Coords. of vertex</td></tr>
<tr><td>5</td><td>lf</td><td>0,1</td><td>Locked / unlocked flag for vertex (optional). 0=unlocked, 1=locked. Repeat fields 2-5 nv times.</td></tr>
</table>

<table>
<tr><th>Card Type</th><th colspan="3">TRI</th></tr>
<tr><th>Description</th><td colspan="3">Lists the triangles in the TIN</td></tr>
<tr><th>Required</th><td colspan="3">NO (a set of triangles can be generated from the vertices)</td></tr>
<tr><th>Format</th><td colspan="3">TRI nt
v_{11} v_{12} v_{13}
v_{21} v_{23} v_{23}
.
.
v_{nt1} v_{nt2} v_{nt3}</td></tr>
<tr><th>Sample</th><td colspan="3"><code>TRI 4</code>
<code>5 1 4</code>
<code>4 1 2</code>
<code>4 2 3</code>
<code>5 4 3</code></td></tr>
<tr><th>Field</th><th>Variable</th><th>Value</th><th>Description</th></tr>
<tr><td>1</td><td>nt</td><td>+</td><td>The number of triangles in the TIN.</td></tr>
<tr><td>2–4</td><td>v1,v2,v3</td><td>+</td><td>Vertices of triangle listed in a counter-clockwise order. Repeat nt times.</td></tr>
</table>

Card Type	ENDT
Card ID	3000
Description	Marks the end of a group of cards describing a TIN. There should be a corresponding BEGT card at a previous point in the file. No fields.
Required	YES

XY Series Files

XY Series Files (*.xys)

The XY Series Editor is used to define x, y series data curves. These data curves can then be used to define such things as:

- Time dependent boundary conditions
- Rating curves

XY Series Files

XY Series files are imported and exported through the XY Series Editor.

XYS Format

```
Line 1  – XYS file identifier, Curve ID, Number of Points, and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XYS 1 5 Head

0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY1 Format – Discontinued Format

Both the x and y values are listed for each point on the curve. There is no limit to the spacing or interval used between subsequent x values.

```
Line 1  – XY1 file identifier, Curve ID, Number of Points, Delta X (not
 used), Delta Y (not used), Repeat (not used), Begin X Cycle (not
used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY1 1 5 0 0 0 0 Head	
0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY2 Format – Discontinued Format

Identical to the XY1 card except that the number of points and the x values are assumed to be static and cannot be altered by the user.

```
Line 1  – XY2 file identifier, Curve ID, Number of Points, Delta X (not
 used), Delta Y (not used), Repeat (not used), Begin X Cycle (not
used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY2 1 5 0 0 0 0 Head	
0.0	0.0
1.0	2.0
2.0	7.0
3.0	8.0
4.0	9.5

XY3 Format – Discontinued Format

The x values are defined by a beginning x value, an increment in x, and a percent change in x per increment (applied after adding the increment). Only the y values are explicitly listed. The x-values are calculated by starting with the Initial X value given and recursively adding the X Increment and then multiplying by the X Percent Change.

```
Line 1  – XY3 file identifier, Curve ID, Number of Points, Initial X, X
 Increment, X Percent Change, Delta X (not used), Delta Y  (not used),
Repeat (not used), Begin X Cycle (not used), and Curve Name.
Line 2+ – X Value, Y Value (one pair per line)
```

Sample File

XY3 1 5 0.0 2 10.0 0 0 0 0 Head

0.0

2.0

7.0

8.0

9.5

Related Topics

File Formats

XY Series Editor

The XY Series Editor is a special dialog that is used to generate and edit curves defined by a list of x and y coordinates. The curve can be created and edited by directly editing the xy coordinates using a spreadsheet list of the coordinates. An entire list of curves can be generated and edited with the Editor and curves can be imported from and exported to text files for future use. You can also paste your xy data directly to the spreadsheet.

The XY Series Editor is used in GMS, SMS, and WMS. It was designed to be general in nature so that it could be used anywhere that a curve or function needs to be defined. In some cases, the x values of the curve must correspond to a pre-defined set of values. For example, the x values may correspond to a set of time steps whose interval is established in a separate dialog. In such cases, the x fields cannot be edited but the y values associated with the pre-defined x values can be edited. In other cases, there is no limit on the number of x values or on the x spacing and both the x and y values can be edited.

The XY Edit Fields

The two vertical columns of edit fields on the left side of the dialog are for direct editing of the xy series values. A pair of application specific titles appears at the top of the columns.

The buttons below the xy edit fields are used to manipulate the values in the edit fields. The buttons are as follows:

Use dates/times For selected situations such as entering time series data in the Map module, it is useful to enter the data in date/time format. Checking this toggle allows the x values in your curve to be entered in date/time format.

Import/Export Buttons The Import and Export buttons allow the user to read in or save an xy series file.

The XY Series Plot

The window in the upper right hand corner of the XY Series Editor is used to plot the curve corresponding to the xy values in the edit fields. As each value in the edit fields is edited, the corresponding point on the curve is adjusted instantaneously. Plot options are accessed by right-clicking on the plot.

Related Pages

- GMS main page
- SMS main page
- WMS main page

XYZ Files

XYZ Files (*.xyz)

The File Import Wizard is used to import delimited or fixed width files. See the File Import Wizard article for more information. XYZ files are opened through the Text Import Wizard. Any ASCII tabular file can be opened through the wizard by renaming the file to have a *.xyz file extension. XYZ files may have an optional file header describing the data in each column of the file. The columns of data follow the header and may be delimited by any character (such as TAB, SPACE, COMMA, etc.). XYZ files are opened using the File | Open menu command.

Related Topics

File Import Wizard File Formats

Generic Vector/Raster Files

There are several kinds of vector files and many kinds of raster files that are automatically identified and read into SMS.

These kinds of vector files include:

- DXF/DWG
- ESRI Shapefiles – read into GIS module
- MapInfo MIF/MID files – read into GIS module

Raster data is read in as images or rasters (for example DEMs). SMS will recognize most image and DEM formats.

The generic vector/raster files can be used to read in vector data not natively supported by SMS or not recognized as a raster/image format. Data unrecognized by SMS should try to load the data as generic vector/raster data. If this fails, Use the "Open As" command and specify for the type "Generic vector/raster data."

Generic vector data is converted into one of the natively supported types chosen by the user. After the conversion has been made, SMS will link to the converted data and not the original file. The conversion is performed by "GlobalMapper from Blue Marble Geographics." All of the formats supported by this package are supported. The list of formats can be found here: http://www.globalmapper.com/product/formats.htm [1].

Import from Web

Overview

The *Import from Web* feature allows you to connect to the internet to download free data – images, elevation data etc. If you have an internet connection, this is an easy and convenient way to acquire this type of data.

The data is made available for free by various entities who provide web services [1]. Each XMS programs has a number of available data types they can retrieve.

The *Import from Web* feature is no longer is use for GMS as of GMS 9.0. The feature is still used in GMS 8.3 and earlier.

GMS	SMS	WMS
• NED data – USGS [2] • ASTER and SRTM data – USGS & NASA [4] • TerraServer aerial photo [3] • TerraServer urban [3] • TerraServer topo [3]	• TerraServer aerial photo [3] • TerraServer urban [3] • TerraServer topo [3]	• NED data – USGS [2] • ASTER and SRTM data – USGS & NASA [4] • NLCD and CORINE (European) Land Cover data [5] • World Imagery [6] • World Street Maps [7] • World Topo Maps [8] • MapQuest OpenStreetMap Worldwide Street Maps [9] • Land Use Shapefiles [10] • STATSGO and SSURGO Soil Type Shapefiles [11] • Harmonized World Soil Database v 1.1 [12] • Global Land Cover [13]

Data Availability

Elevation (NED, ASTER, and SRTM) Data

- NED data contains the best available raster elevation data of the conterminous United States, Alaska, Hawaii, and territorial islands. NED data are not available for other areas.
- ASTER and SRTM data are available for most of the earth's surface. The ASTER dataset is reliable and high-quality.

Imagery

All the imagery (World Imagery, Street Maps, Topo Maps, and OpenStreetMap.org data) are available for anywhere on the earth. Besides downloading these images using the Import from Web command, these images can be read as online maps and then converted to a static map (stored locally on your computer) for faster display after you have decided on a project area.

Land Cover Data

- The 100 m Resolution CORINE dataset (raster) is available for anywhere in Europe.
- The 30 m NLCD dataset (raster) is only available for the conterminous United States.
- The Land Use Shapefile dataset is currently only available for the states of Utah and Louisiana.
- The Global Land Cover dataset is currently only available for Morocco. If you are interested in making data available for your region, you can either follow the following steps or contact Aquaveo, who can make data for your area available on their web servers for a fee. Follow the following steps to bring Land Use data for your area into WMS:

1. Go to the European Space Agency site to download land use data [13].
2. Download the .zip file Globcover2009_V2.3_Global_.zip and unzip this file on your computer.
3. Open GLOBCOVER_L4_200901_200912_V2.3.tif in a GIS (such as ArcMap) and convert it to an ESRI raster file. Trim the raster as needed, then convert the raster to a shapefile.
4. Convert the file Globcover2009_Legend.xls to a *.dbf file and join this file with the shapefile values to get the land use names and IDs.

Soil Data

- SSURGO soil datasets are available for some parts of Utah, Colorado, and California and are available for most of Louisiana.
- A STATSGO soil dataset is available for all of Utah. You can also download the STATSGO dataset for the entire United States, but this is a large download (338 MB) and will take some time to download.
- Data from the Harmonized World Soil Database are only available for Morocco. If you are interested in making data available for your region, you can either follow the following steps or contact Aquaveo, who can make data for your area available on their web servers for a fee. Follow the following steps to bring soil data for your area into WMS:

1. Download and install the Harmonized World Soil Database program to your computer from the Harmonized World Soil Database web site [14].
2. Launch the HWSD Viewer on your computer. The soil data will be copied to the folder c:\program files (x86)\HWSD_v<xxx>\Data where <xxx> is the version of the viewer you have downloaded. The program may also be installed in c:\program files\<...> if you are running a 32-bit version of Windows. The following files are contained in this folder:
    1. The HWSD Raster *.zip file.
    2. The HWSD DBF file.
    3. The HWSD_META DBF file.
3. Copy the files in the data folder to a writable location on your computer and unzip the HWSD Raster *.zip file.
4. Open the .bil file in ArcMap and convert the *.bil file to a shapefile using the IDs.
5. Join the HWSD DBF file with the IDs in the shapefile.
6. Join the attribute IDs with the HWSD_META DBF file. This gives you a shapefile with the soil IDs and various soil attributes that can be used for hydrologic modeling in WMS.

Additional Information

Note that more vector-based soil and land use datasets are available; Contact Aquaveo [15] if you are interested in adding data from a specific area to the list of available land use or soil data that can be downloaded. A comprehensive list of soil and land use data available for download is located here.

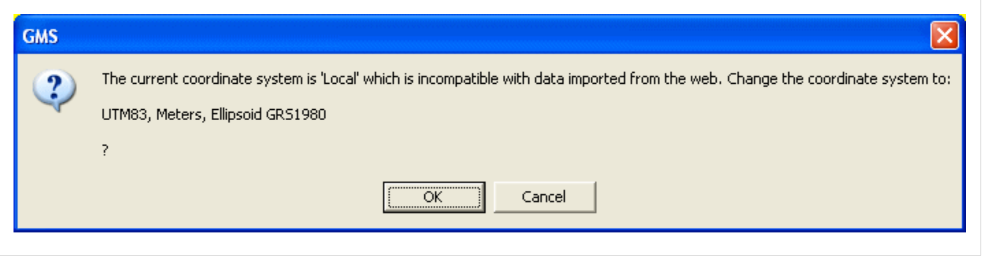
Terraserver images are no longer available because this web service has gone offline.

Using the Import from Web Command

This feature is found via the ***File|Import from Web*** command. This command results in the following series of dialogs.

1. **Warning**

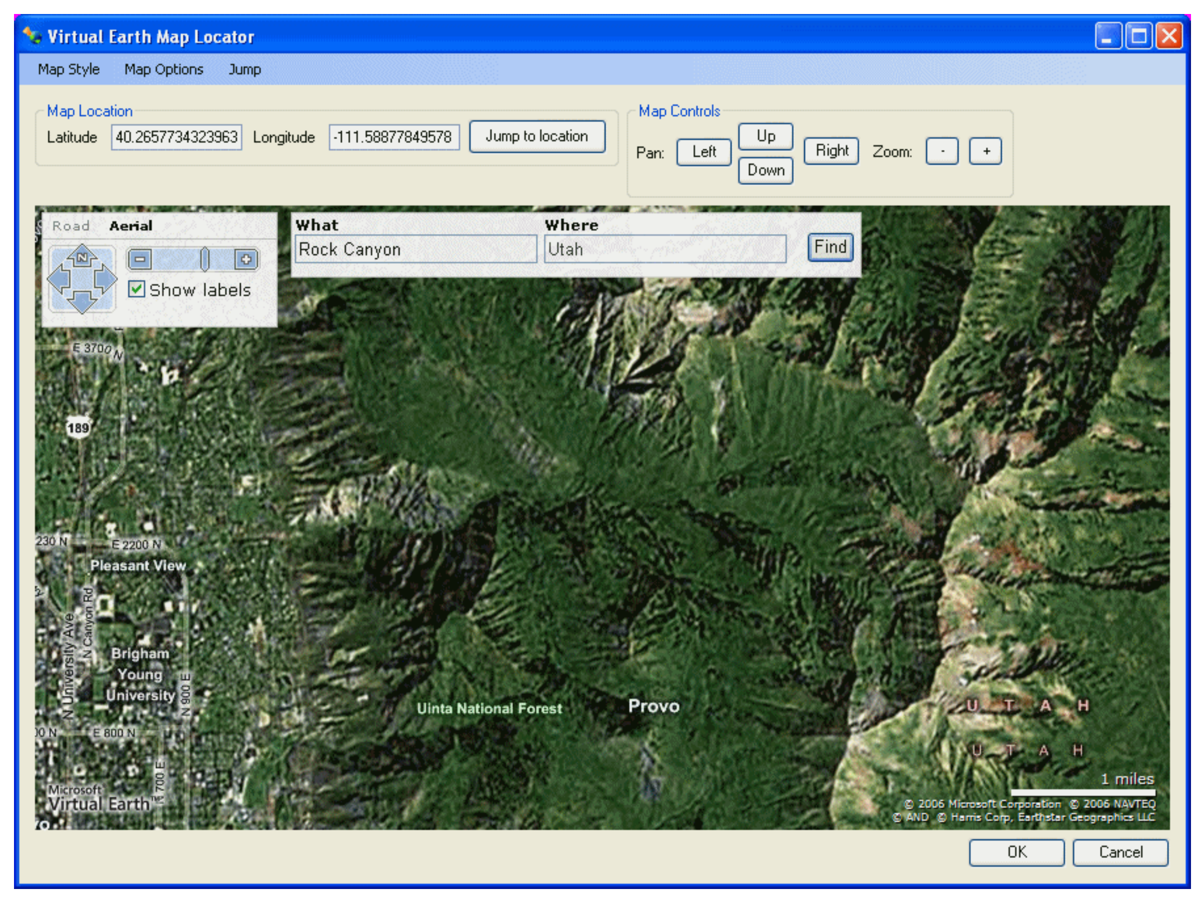
   If your current coordinate system is "Local", you will get the following warning dialog. The data that will be downloaded is georeferenced so your project coordinate system must be something other than 'Local'. Selecting **OK** will change the current coordinate system.



2. **Virtual Earth Map Locator**

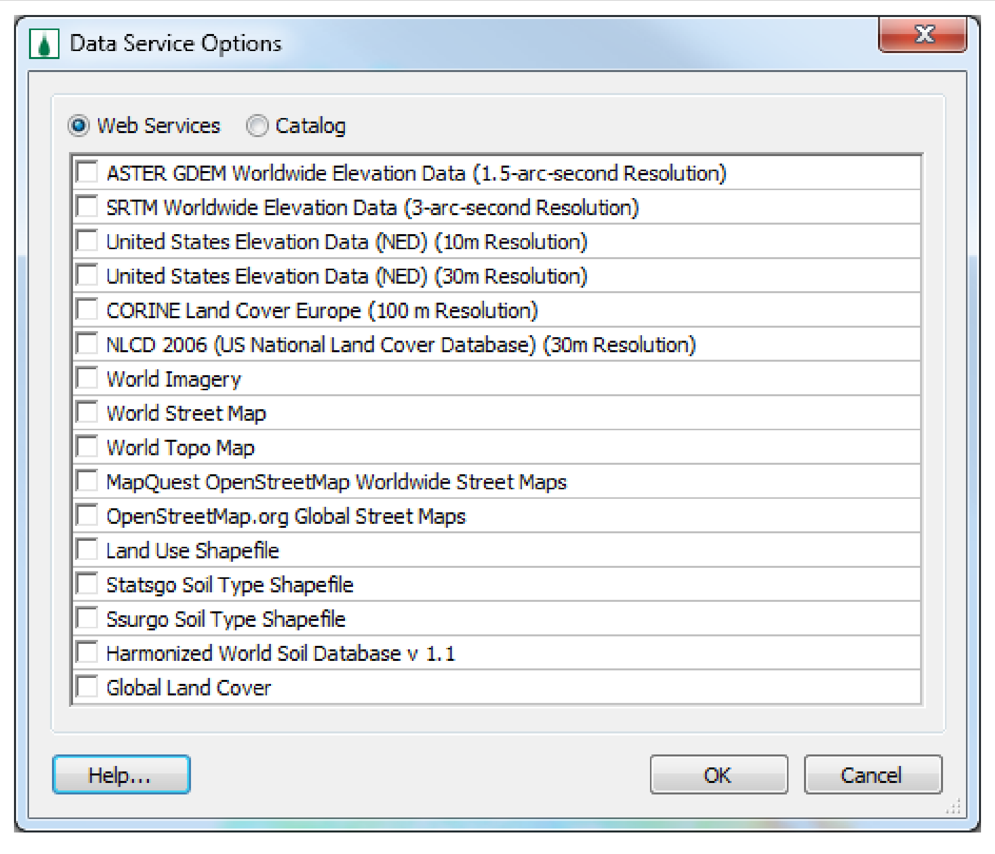
   Use the map in this dialog to go to your location of interest.

- You can **zoom** in or out using the controls or the mouse wheel.
- You can **pan** using the controls or by clicking and dragging. You can also enter the latitude and longitude to jump to a specific location.
- Use the *Map Options* menu to turn on the floating controls in the map (search, pan and zoom).
- Use the *Map Style* menu, or the floating controls, to change the map between **Road**, **Aerial**, and **Hybrid**.



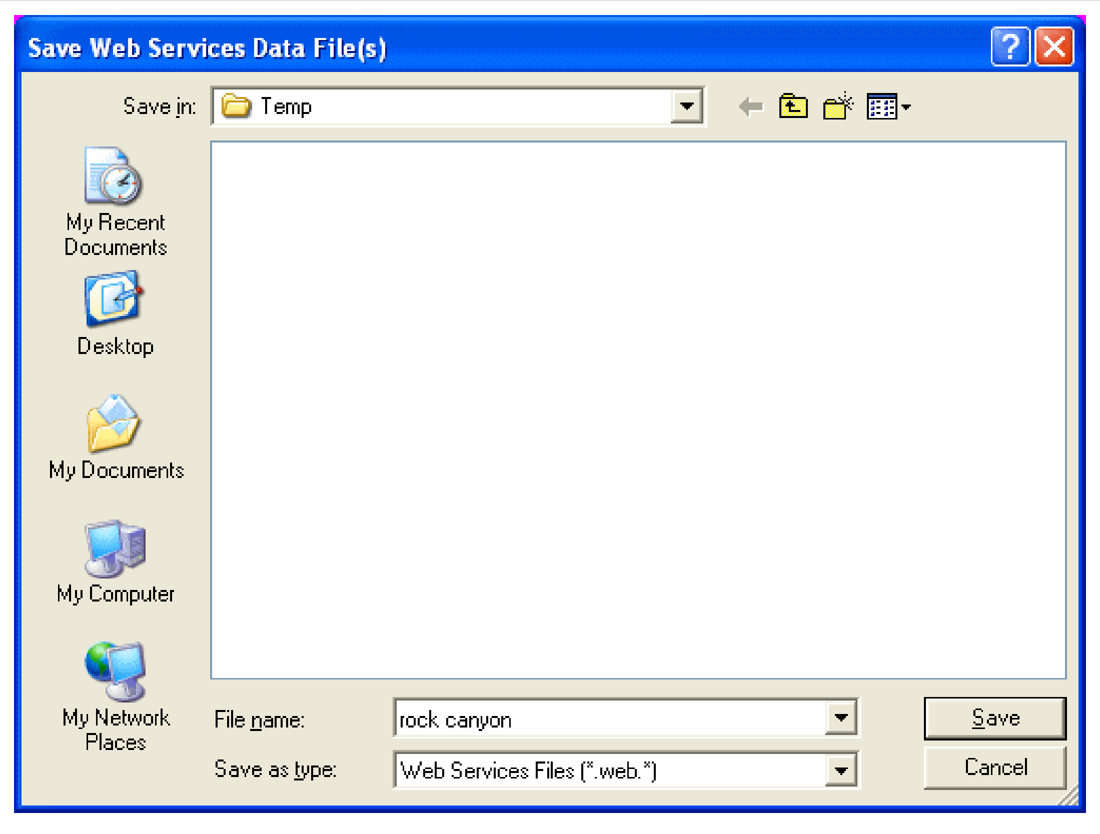
3. **Data Service Options**

   Here you select which type of data you are interested in.



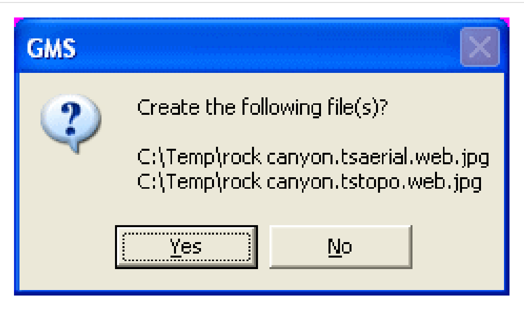
4. **Save**

   Now you are asked where you want to save the data. You only need to specify one file name, even if you've selected more than one type of data in the previous dialog. The files will all be given the same prefix but different suffixes.



5. **Confirm File Creation**

   You may be asked to confirm that you want to create the files.

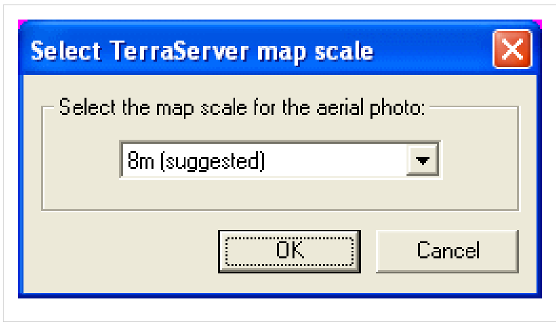


6. **Initialize Connection**

   The following dialog is shown while the connection is being made.

7. **Select Scale**

   Smaller numbers (larger scales) will result in better resolution, but longer download times.



8. **Downloading**

   This dialog reports the download progress. If you click **Abort**, your image will exist but will be only that portion that you've downloaded so far.

Steps 6-8 will repeat for each data type you selected in step 4. After everything is finished, the data (images etc.) will appear in the Project Explorer.

Registering an Image

If an image file is not geo-referenced then the user must define the coordinate system of the image. The register dialog allows the user to specify the coordinate system for the image. When an image is opened, if the image is not self-referenced, XMS attempts to find world file with the same name as the image (*.wld or *.jpgw extension). If neither of these is found, the register dialog opens.

What is Image Registration?

Before an image can be displayed, the image must be "registered" or geo-referenced. Registering an image involves identifying points on the image corresponding to locations with known real world (XY) coordinates. Once these points are identified, they are used to scale and translate the image to the proper location when it is drawn with the other objects in the Graphics Window. If an image is not registered properly, any objects which are created using the background image as a guide will have the wrong coordinates.

Register Image Dialog

An image is registered using the Register Image dialog. The main feature of the Register Image dialog is a large window in which the image is displayed. Two or three points (shown by "+" symbols) are also displayed in the window. These points are used to identify locations with known real world coordinates. The real world coordinates (X,Y) and image coordinates (U,V) of the registration points are listed in edit fields below the image. The points are moved to the desired locations on the image by dragging the points using the tools described below. Once the points are located, the real world coordinates can be entered in the corresponding edit fields. The dialog contains the following options:

- **2 point or 3 point registration** – Two point registration rotates and uniformly scales an image. Three point registration allows for non-uniform scaling to account for some parallax.
- **Import World File** – Used to import a TIFF world file (*.tfw). A TIFF world file has the information needed to set the (X,Y) and (U,V) coordinates in order to place the image in the correct world coordinates.
- **Image name** – Used to associate a name with the file. This name will appear in the project explorer.

Register Image Dialog Tools

The following tools can be used to help position the registration points:

Tool	Tool Name	Description
	Select Point Tool	The Select Point tool is used to select and drag register points to a location on the map for which real coordinates are known so that they can be entered in the corresponding XY edit fields.
	Zoom Tool	In some cases, it is useful to magnify a portion of the image so that a registration point can be placed with more accuracy. The Zoom Tool is used to zoom in a portion of the image.
	Pan Tool	After zooming in on a portion of the image, the Pan Tool is used to pan the image vertically or horizontally.
	Frame Macro	The Frame Macro is used to automatically center the entire image within the drawing window of the dialog after panning and zooming in on a specific location.

Import World File

The **Import World File** button can be used to automatically define the registration data. A world file is a special file associated with a previously registered image that is exported from ArcView® [1] or Arc/Info® [2]. The file contains registration data that can be used to register the image.

Saving/Reading Image Registration Data

When a project file is saved, a link to the image is saved in the project file, along with the current image registration information so that the image is re-registered to the same coordinates every time the project is opened. The original image file and world file (if one exists) are not altered.

Convert Point Coordinate System

The x, y coordinates of each register point must be specified. If the user has the (x,y) coordinates in a different coordinate system than their project, the coordinates will need to be converted.

GMS Point Conversion

The Convert Point button in the image registration dialog will allow the user to convert the coordinates.

SMS Point Conversion

The Single Point Conversion command in the *Edit* menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

An alternative approach is to convert the coordinate system after importing by right clicking on the image in the Project Explorer and choosing ***Coordinate Conversion*** from the right click menu.

WMS Point Conversion

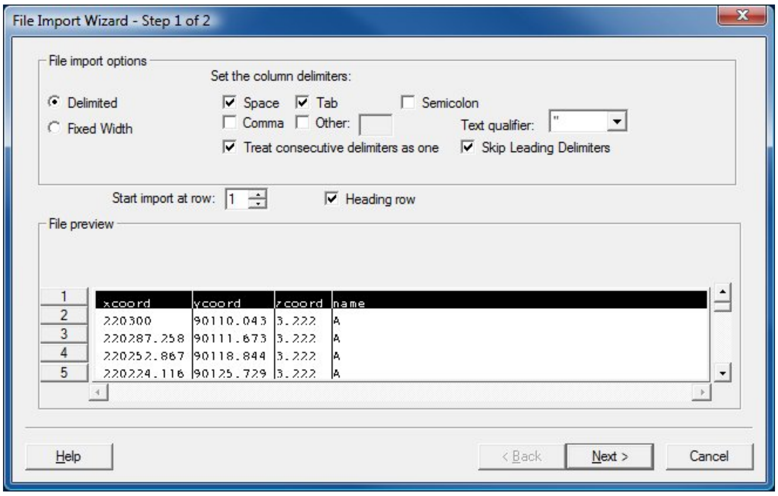
The Single Point Conversion command in the **Edit** menu can be helpful if you need to convert between any two coordinate systems. You should perform this conversion and record the locations in the correct coordinate system prior to entering the registration dialog.

7.2.b. File Import Wizards

File Import Wizard

SMS can import many files generated by other software in their native format. Refer to Importing Non-native SMS Files for a list. For files that are not included in the list, SMS provides the **Text Import Wizard**.

The **Text Import Wizard** enables users to import many different types of data into SMS. The Text Import Wizard is initialized by selecting a *.txt file in the **Open** command from the **File** menu. The wizard has two steps:

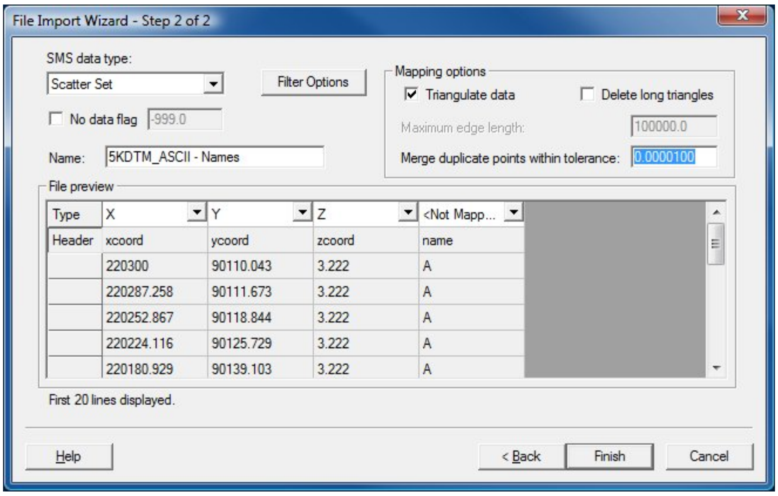


Step 1 of the Text Import Wizard – File import options

Step 1 – Delimiting Columns

The first step in the wizard allows users to delimit the data into columns. The following options exist to delimit the data:

- **Delimited** – For the Delimited option, typical delimiters are included as well as an option for users to specify a delimiter.
- **Fixed Width** – Columns can be specified with a fixed width by clicking on the ruler bar or the window with the data. Break lines can be dragged, and they can be deleted by double-clicking on the break line or dragging them off the screen.



Step 2 of the Text Import Wizard – File format

The user can specify the starting row the data will be imported at. If your data has a row of headings, you can indicate such and SMS will use the headings in the next step to determine what kind of data each column represents.

Step 2 – Assigning Column Types

The first 20 lines of the file are displayed in a spreadsheet according to the file outline specified in step 1. This step lets you pick what kind of data you are importing (see Supported File Formats). A "no data flag" can be specified for the file. This is a number that, when encountered in the file, tells SMS to mark the value as "NULL" or "no data". For example, a water surface elevation dataset would assign a no data flag to dry nodes.

The data in the columns are identified by selecting the type in the combo box at the top of each column in the spreadsheet. If a row of headings exists, SMS will automatically select the proper type if it recognizes the heading. Otherwise they are Not Mapped by default. The available column types changes depending on the SMS data type selected. Certain column types must be mapped for each file format before the user can progress to the next step in the wizard. The name of each column is changed by editing the Header cell.

Mapping Options

When reading in a scatter set or mesh data, the following mapping options are available:

- **Triangulate data** – Triangulates the scatter vertices / mesh nodes
- **Merge duplicate** – Merges duplicate scatter vertices / mesh nodes based on the specified tolerance
- **Delete long triangles** – Deletes scatter triangles with an edge length longer than the specified edge length
- **Append mesh** – Appends the mesh nodes to the existing mesh

Filter Options

When importing a Scatter Set, pressing the "Filter Options" button will open the File Import Filter Options dialog. The filter options are useful when reading scatter sets that are too large for SMS to successfully read in. Once the scatter set has been read into SMS, the more sophisticated normals filtering algorithm can be used.

Additional Options

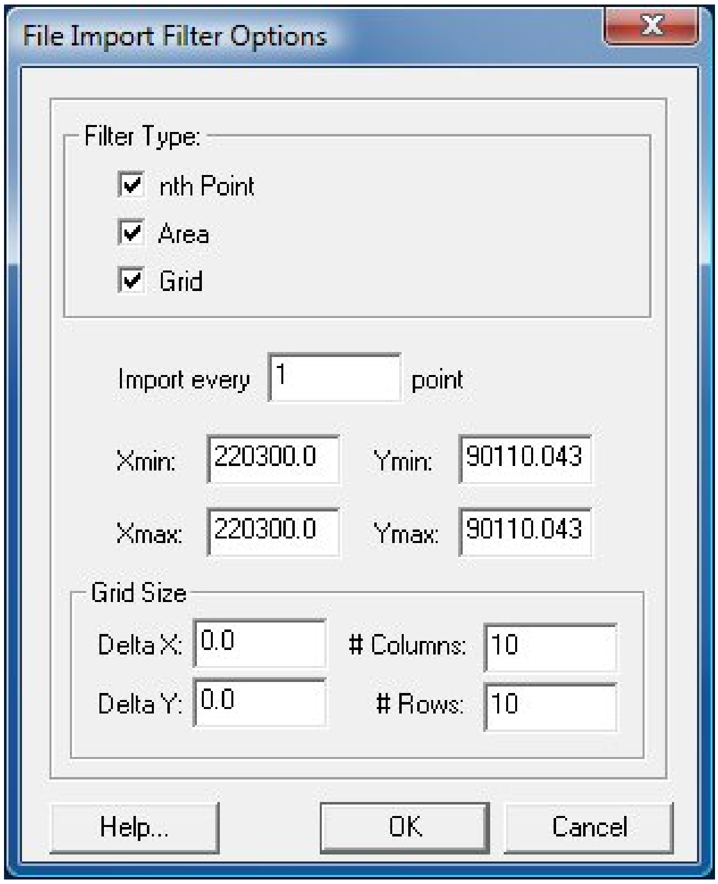
After the data have been imported, the coordinate transformation tools can be used to transform and translate the data.

Related Topics

- File Formats

File Import Filter Options

When importing a Scatter Set using the File Import Wizard, pressing the "Filter Options" button will open the File Import Filter Options dialog. The filter options are useful when reading scatter sets that are too large for SMS to successfully read in. Once the scatter set has been read into SMS, the more sophisticated normals filtering algorithm can be used.



Filter Options

The following filter options are available:

- **nth Point** – Simple method to reduce the scatter set size by reading a reduced number of vertices from the file. Reading every 2nd point will result in a 50% reduction in vertices, every 4th point will result in a 75% reduction, etc.
- **Area** – Only reads points falling within the specified x, y boundary. Useful for filtering data outside of the area of interest.
- **Grid** – Scatter vertices are created on a user defined grid. Each vertex has a "bucket" around it. The z-value is assigned to the vertex based on the average value of the vertices in the "bucket."

Related Topics

- File Import Wizard
- Normals Filtering

File Import Wizard Supported File Formats

The following types of data can be imported into SMS via the File Import Wizard.

- 2D Scatter Set Vertices
- 2D Mesh Nodes
- Feature Points
- Observation Data
- Wind, Wave, Water level

A description of the fields (columns) that SMS recognizes when importing text files is provided in the tables below.

2D Scatter Vertices

Field	Type	Required	Comments
X	Number	yes	X-location
Y	Number	yes	Y-location
Pt Name	Text	no	
Vector X	Number	no	Used in conjunction with Vector Y Field
Vector Y	Number	no	Used in conjunction with Vector X Field
Vector Magnitude	Number	no	Used in conjunction with Vector Direction Field
Vector Direction	Number	no	Used in conjunction with Vector Magnitude Field
Scalar Data	Number	no	
Breakline	Text or Number	no	See Scatter Breakline Options for a discussion of Breakline Delimeters

"id"	"x"	"y"	"xylene"	"toluene 0.0"	"toluene 2.0"
"OW-21"	32.4	5234.3	300	999	999
"0W-22"	93.4	5832.3	84	398	401
"0W-23"	83.3	8438.2	89	47	52

2D Mesh Nodes

Field	Type	Required	Comments
X	Number	yes	X-location
Y	Number	yes	Y-location
Z	Number	yes	Z-location

"x"	"y"	"z"
32.4	5234.3	12.34
93.4	5832.3	13.47
83.3	8438.2	21.54

Feature Points

Field	Type	Required	Comments
Name	Text	no	
X	Number	yes	X-location
Y	Number	yes	Y-location
Z	Number	yes	Z-location

"name"	"x"	"y"	"z"
"Pt. 1"	32.4	5234.3	12.34
"Pt. 2"	93.4	5832.3	13.47
"Pt. 3"	83.3	8438.2	21.54

Observation Points

Field	Type	Required	Comments
Point Name	Text	no	
X	Number	yes	
Y	Number	yes	
Z	Number	no	
Measurement	Text	no	Measurement name. Multiple measurements allowed.
Interval	Number	no	

"name"	"x"	"y"	"z"	"hd"	"int"
"OBS_Q5"	23.3	44.2	32.2	567.5	1.2
"OBS_Q6"	83.3	84.3	32.2	555.3	1.4
"OBS_Q7"	85.3	39.3	33.2	999	0

PTM Trap Output Data

Field	Type	Required	Comments
Step	Number	no	Time step index of an entry event for a trap
Date	####/##/##	yes	Date of an entry event - must have a year, month and day
Time	##:##:##.####	yes	This column must have hour, minute and second of the entry event
Particle	Number	no	This column could be used to reference other PTM output files. Not used in import wizard
Trap	Number	yes	Defines which trap this parcel entered
Value column	Number	no	This is an optional column. There may be more than one.
Filter column	Number	no	This is an optional column. There may be more than one.

See PTM Trap Output for more information on reading PTM trap output files.

Wind, Wave, Water level

Field	Type	Required	Comments
Date/Time	Number/Number	yes	
Date	Number	yes	
Time	Number	yes	
Primary Height	Number	yes	Used in conjunction with Primary Period & Direction Field
Primary Period	Number	yes	Used in conjunction with Primary Height & Direction Field
Primary Direction	Number	yes	Used in conjunction with Primary Height & Period Field
Secondary Height	Number	no	Used in conjunction with Secondary Period & Direction Field
Secondary Period	Number	no	Used in conjunction with Secondary Height & Direction Field
Secondary Direction	Number	no	Used in conjunction with Secondary Height & Period Field

7.2.c. GSDA

7.2.c.1. Digital Elevation

GSDA:Digital Elevation

Topography and Terrain

Obtain DEMs directly from WMS using the Get Data tools [1]

WMS offers options to download various types of digital elevation data for anywhere in the world using the Get Data Toolbar. Use either the Get Data From Map button to select an area and download data for the selected area or the Get Data tool to get data for a selected area in the WMS window. This is the recommended method to get DEM data into WMS unless you have other DEM sources that may be more recent or of better quality.

Obtain Data from USGS Seamless Map [2]

The USGS provides DEM datasets as part of the Seamless Data Distribution System (SDDS). The two types of data provided as part of this system include:

Shuttle Radar Topography Mission

30 Meter - Data available for contiguous US, Hawaii and southern Alaska

90 Meter - Extensive coverage including South America and most of North America

National Elevation Dataset (NED)

1/9 Arc Second - 3 meter resolution limited to select areas of Washington state

1/3 Arc Second - 10 meter res. available covering most of the contiguous US

1 Arc Second - 30 meter res. for all of the United States

2 Arc Second - Dataset specific for the state of Alaska

Details:

SRTM and NED data is offered in Gridfloat, BIL, ArcGRID, and TIFF formats. (To download a DEM, select the Gridfloat type)

You can define a custom, seamless area to download.

The interface allows you to display various GIS layers to aid in the selection process.

Data can be downloaded in files up to 100MB maximum size.

Downloads are free (no cost).

Download various data (including elevation) from the USGS [3]

Details:

Download DEM data in various formats.

Download various land cover and vegetation data.

Download USGS imagery.

Downloads are free, but an account must be created to download.

Download various data collected by NASA [4]

Details:

Download DEM data from various sources and in various formats from around the world.

Downloads are free, but an account must be created to download.

DEM Data from webGIS [5]

Details:

Terrain data offered

USGS 7.5 minutes - 1:24K scale (~30 m res) <Covers all of contiguous United States>

USGS 1 degree - 1:250K scale (~90 m res) <Hawaii and Alaska only>

Data only available in *.dem format

Graphical US map makes locating an area easy

Downloads are free (no cost).

Canadian DEM Data [6]

Worldwide DEM data in SRTM format-90 meter resolution [7]

SDTS DEM data from MapMart [8]

Details:

Terrain data offered

10 Meter SDTS (Spatial Data Transfer Standard) <Limited coverage of US>

30 Meter SDTS <Covers all of contiguous United States and Hawaii>

Limit of 10 quads per download

Graphical US map makes locating an area easy

Downloads are free (no cost).

The following types of formats are also available for a fee

USGS DEM

XVY ASCII Grid

Arc ASCII Grid

DXF Mesh

SDTS DEM Data from GIS Data Depot [9]

Details:

Digital Elevation Model (DEM) data:

10 Meter Resolution

30 Meter Resolution

SDTS Format

Extensive coverage of the U.S.

Free downloads.

Data offered free may also be available in other formats for a fee.

Tips:

The USGS GNIS [10] and Google Maps [11] are useful tools to determine the *name* of the DEM you need.

DEM Data from Land Info [12]

Details:

Digital Elevation Model (DEM) data:

1:250K Resolution

1:24K Resolution

Numerous formats

*.dem

ESRI GRID

ESRI BIL w/HDR

*.dted

Downloads must be purchased

Bathymetric

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_NGDC.png [13] Obtain bathymetry data using GEODAS Design-a-Grid [14]

The National Oceanic and Atmospheric Administration (NOAA) has produced the National Geophysical Data Center (NGDC) website to allow for US data access. Oceanic bathymetry can be found using GEODAS (GEOphysical DAta System) Grid Translator Design-a-Grid.

Details:

- Must indicate latitude and longitude degree boundaries
- Three format choices all available with headers:

1. Binary Raster
2. ASCII Raster
3. XYZ Delimited

- Downloads are free (no cost).

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_NOS.png [13] Obtain estuarine bathymetry data from NOAA's NOS [15]

The National Oceanic and Atmospheric Association (NOAA) has produced the National Ocean Service (NOS) Estuarine Bathymetry website to allow for US estuarine bathymetry access.

Details:

- Estuaries divided into six regions:

1. North Atlantic
2. Middle Atlantic
3. South Atlantic
4. East Coast Gulf of Mexico
5. West Coast Gulf of Mexico
6. Pacific Coast

- 4 downloadable options of individual estuaries in *.dem format:

1. 1 Arc Second (30 meter resolution) - Split into 7.5" quads
2. 1 Arc Second - One big file
3. 3 Arc Seconds (90 meter resolution) - Split into 1 degree quads
4. 3 Arc Seconds - One big file

- Downloads are free (no cost).

Obtain topographic and bathymetric data from the Topographic and Bathymetric Data Inventory [16]

The National Oceanic and Atmospheric Association (NOAA) offers topographic and bathymetric data from the NOAA coastal Services Center and the Federal emergency Management Agency through the Topographic and Bathymetric Data Inventory. This resource provides both topographic and bathymetric data for the coastal areas of the United States. More information on this resource can be found here [17].

DEM Overview

There are many uses and applications of DEMs. GIS (Geographic Information System) software such as ArcView and ARC/INFO, as well as modeling software such as WMS (Watershed Modeling System) can use DEMs for many engineering and scientific applications. WMS uses DEM data to produce watersheds which are then used to model storm events, create hydrographs, route floods down rivers and through reservoirs, etc. This information can be used to design culverts, dams, detention basins and other hydraulic structures. DEM data are commonly used to create another type of digital terrain model called a TIN (Triangulated Irregular Network). Public domain software is available for tasks such as simply viewing a DEM -- one example is dlgv32 Pro [18], a tool provided by the USGS.

With powerful modeling software such as WMS, using DEM data to run a model is not difficult, and can be accomplished in four general steps:

1. Find and/or download the needed DEM data.
2. Import the DEM(s) into WMS
3. Delineate the watershed by inserting stream networks, one or more outlets, and reservoirs. If desired, a TIN can be created as well. Additional hydrologic data such as land use and soil type can also be used in WMS.
4. Run the model and view the results. WMS supports several models such as HEC-1, NFF, Rational, TR-55, TR-20, HEC-HMS, and GSSHA.

A DEM (Digital Elevation Model) is simply a digital map of elevation data. These maps, a type of DTM (Digital Terrain Model), are raster data meaning that they are made up of equally sized gridded cells each with a unique elevation.

DEMs come in different scales and resolutions. For example, 1:24,000 scale DEM is simply a USGS (United States Geological Survey) 7.5' quadrangle that has been digitized and each cell in the DEM represents a block of terrain 30 meters x 30 meters. The 1:250,000 scale DEM (also known as a 1-degree or a 3 arc-second DEM) has a resolution of 90 meters x 90 meters. DEMs with better resolution are available, but require large amounts of computer memory and disk space and are often impractical to use for large areas of land. If an individual DEM does not cover the entire area of interest, then multiple DEMs can be tiled together to make one large DEM.

The projection and datum for a DEM varies. A common projection for DEMs is UTM (Universal Transverse Mercator) coordinates (meters) and have a specific datum associated with them. Elevations are usually in meters, but sometimes are in feet for areas of low relief, and are referenced to mean sea level.

DEM Tips

- How do I obtain a DEM from USGS?
- How do I obtain a DEM from GIS Data Depot?
- How do I use the USGS Map Locator?
- How do I use the USGS GNIS (Geographic Names Information System)?
- How do I decompress data files?
- How do I import a DEM (native *.dem or SDTS format) into WMS?
- How do I import an NED Gridfloat file into WMS?
- How do I use WMS to convert data to a different coordinate system?

Bathymetry Tips

- How do I obtain bathymetry data from GEODAS Design-a-Grid?
- How do I obtain estuarine bathymetry data from NOAA's NOS?
- How do I import bathymetry data into SMS?
- How do I import estuarine bathymetry data into SMS?

References

[1] http://www.xmswiki.com/xms/Import_from_Web
[2] http://seamless.usgs.gov/website/seamless/viewer.htm
[3] http://earthexplorer.usgs.gov/
[4] http://demex.cr.usgs.gov/DEMEX/
[5] http://www.webgis.com/
[6] http://www.geobase.ca/
[7] http://srtm.csi.cgiar.org/
[8] http://www.mapmart.com/DEM/DEM.htm
[9] http://data.geocomm.com/dem/demdownload.html
[10] http://geonames.usgs.gov/
[11] http://maps.google.com
[12] http://www.landinfo.com/
[13] http://www.noaa.gov/
[14] http://www.ngdc.noaa.gov/mgg/gdas/gd_designagrid.html
[15] http://estuarinebathymetry.noaa.gov/
[16] http://www.csc.noaa.gov/topobathy/#
[17] http://www.csc.noaa.gov/digitalcoast/tools/topobathy/index.html
[18] http://mcmcweb.er.usgs.gov/drc/dlgv32pro/

GSDA:Decompressing Files

Background

Most compressed files are *.zip files and may have a .zip file extension at the end of the filename. Some compressed files such as 7.5-minute DEM files are downloaded as a *.tar.gz file type, and the 1:250,000 DEMs are downloaded as *.gz files. Compressed files must be decompressed before using them in WMS or elsewhere. Decompression steps are slightly different for different types of compressed files. The general steps for decompressing any of these files are:

1. Download an decompression utility (if you haven't already).
2. Decompressing the file(s).

Downloading a Decompression Utility

You can download an evaluation version of WinZip (www.winzip.com [1]) or other Public Domain zip / unzip programs to decompress your files. Try using any popular internet search engine to find other decompression software (a powerful, free decompression program is ZipGenius, which can be downloaded from www.zipgenius.it).

Decompressing a Typical *.zip File

WinZip

- Find the compressed file.
- Either start WinZip and open your file, or simply double-click on the compressed file and WinZip should start automatically. The file(s) will be shown in the WinZip window.
- Select Extract.
- Choose the destination folder and select Extract again. The decompressed file(s) should now be in the destination folder.

The Quick Method

- Right click on the compressed file and choose Extract to folder (followed by the current path).

The decompressed files are extracted to a folder with the same name as your compressed file, located in the same directory as the compressed file. You may need to refresh the directory view in order to see the new folder.

Decompressing and Untarring a *.gz or *.tgz file

You can decompress a *.gz or *.tgz compressed file with WinZip or with the "GZIP and Tar" utility -- the procedures for each method are as follows::

WinZip

The file name should have a file extension of *.gz or *.tgz. If not, rename it accordingly,

- Double click on the compressed file to Start WinZip,
- Choose YES when asked "should WinZip decompress it to a temporary folder and open it?"
- Click the Extract button at the top,
- Enter a location in the "extract to" field where you want to save the extracted files,
- Click the Extract button,

The Quick Method

- Right click on the compressed file and choose Extract to folder (followed by the current path),
- Choose YES when asked "should WinZip decompress it to a temporary folder and open it?"

The decompressed files are extracted to a folder with the same name as your compressed file, located in the same directory as the compressed file. You may need to refresh the directory view in order to see the new folder.

Gzip and Tar

- Rename the file to remove the numbers and periods. The file name can only have 8 characters in the name and 3 characters in the suffix. The file should be <filename.gz>
- Open a MS DOS window.
- Go to the directory where you saved the programs gzip.exe and tar.exe. Your DEM file needs to be in this same directory.
- To decompress the file type... <gzip -d filename>.
- To untar the file type... < tar -xvf filename>.
- You should have about 18 files, most with the .ddf file extension.

References

[1] http://www.winzip.com

GSDA:Importing Bathymetry Data into SMS

1. Open SMS.
2. Got to the File menu and select Open.
3. In the Open window browse to where you unzipped your xyz file.

   NOTE: If your bathymetry data from GEODAS were saved as ps3.zip a folder labeled ps3_data would contain your unzipped data. The xyz file would be in an additional folder labeled ps3 inside the ps3_data folder.
4. Select the xyz file and click Open. The file import wizard will initialize.
5. Check the settings, the defaults should be correct. Click the Next > button.
6. On this page the SMS data type should be set to scatter set and the data columns should be mapped out. If not set them to X, Y, and Z respectively.
7. Click the Finish button. Wait for the file to be loaded.

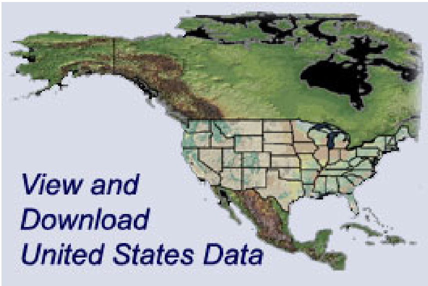
   NOTE: The map will appear as solid colors. For better viewing adjust display settings, i.e.; turn off points and turn on contours.

The bathymetry data has now been imported into SMS for modeling work.

GSDA:Navigating the USGS Seamless Website

Navigating the Seamless Map

The majority of digital elevation data available from the USGS is United States, but also includes other North American countries. For the sake of simplicity select the **View and Download United States Data** section.



This will bring up the Seamless Data Distribution viewer, with North America in the viewer window.

The webpage is laid out in four sections:

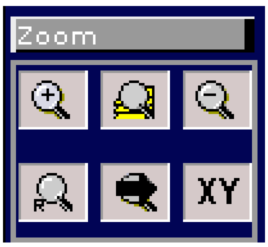
1. Tools
2. Viewer Window (MAP)
3. Scale Information
4. Layers

Tools

The tools are found on the left-hand side of the webpage and are divided into various groups.

- Zoom
- Query
- Tools
- Downloads
- Documents

ZOOM Tools



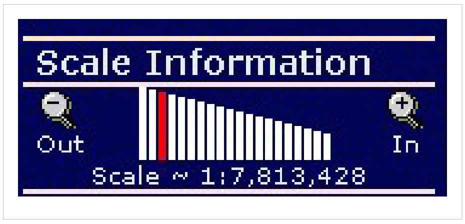
There are three basic Zoom tools which are easily understood.

-  - Zoom In
-  - Zoom to Full Extent
-  - Zoom Out

When using the Zoom In/Out tools it is possible to click either a single point or select a boxed area.

Defining an area will zoom in/out a greater degree than if merely clicking on a point.

Clicking a point will change the scale factor of the map a single degree as shown in the scale information box on the upper-right-hand side of the page.



There are an additional three Zoom tools.

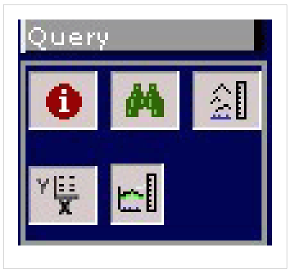
- - Zoom to Region or Area
- - Zoom to Previous Extent
- - Zoom to a Point

Selecting the *Zoom to Region or Area* tool will prompt input boxes below the viewer window. Simply choose whichever selection you desire and the map will immediately update to that location.

Selecting the *Zoom to Previous Extent* will just take you back a step.

Selecting the *Zoom to a Point* will open an Explorer User Prompt at the top of the screen, prompting you to input a coordinate in longitude, latitude degrees.

QUERY Tools



There are a number of Query tools available.

- - Displays the Map Feature Attributes
- - Search for a Geographical Place
- - Elevation Query
- - US Grid National Query
- - Elevation Profile

Selecting the *Display the Map Feature Attributes* and clicking on a point will open an OGC Identify Features window displaying county and state information.

Selecting the *Search for a Geographical Place* will open an Explorer User Prompt at the top of the screen, inquiring for a known place. Once entered search results will be visible in a list below the map.

Selecting the *Elevation Query* tool allows you to choose a point on the map and it will display the elevation above sea-level, latitude and longitude coordinates, as well as the data source below the map.

Selecting the *US National Grid Query* tool and picking a point on the map gives the USNG Coordinate number and latitude, longitude coordinates below the map.

Selecting the *Elevation Profile* icon will allow you to select points on the map and generate elevation profiles between a minimum of two points. More points can be used to create more profiles. The profile can also be generated in either units of feet or meters. Greater detail can be obtained from a greater sample/segment number.

TOOLS Tools

There are a number of Tools tools available.

- - Pan
- - Measure
- - Clear Measure Totals
- - Create PDF Image
- - Metadata
- - Layer Transparency

The *Pan* tool is the regular movement of the map.

The *Measure* tool allows you to tally distance in miles from one selected point to another. Besides indicating the current segment's length is also keeps a running total distance of all segments constructed. This information is kept in boxes at the top of the map window.

The *Clear Measure Totals* simply resets the Total and Segment distance boxes at zero.

The *Create PDF Image* icon opens a Create PDF Image window where the image can be given a title, the page layout and image size can be decided. Once finished the Get Image button can be clicked to go to the Request Status window. After a short while the image is ready and can be saved to a disk.

The *Metadata* tool opens a dropdown list of metadata layers available below the viewer window. Select which one you want then click on the desired location on the map to have an extraction window of the metadata open up.

The *Layer Transparency* tool opens a dropdown list, under the map window, of active display layers which can have their transparency changed. When you click the Apply Transparency button the map will refresh accordingly.

DOWNLOADS Tools

There are just two Downloads tools to choose from.

- - Define Download Area
- - Define Area by Coordinates

With the *Define Download Area* tool you can select an area for which you would like digital elevation data. Once selected an SDDS Request Summary Page will open. Default data for the area is NED 1 Arc

Second in ArcGRID format. Your request may be adjusted by clicking the Modify Data Request button. You will be taken to the Options Page where you can modify datasets, and formats. Changes can be saved by selecting the appropriate button on the bottom of the Options Page. Click the download button on the Request Summary Page to start data transfer. Wait patiently for the Download Request Status window to finish and save your file to disk.

When using the *Define Area by Coordinates* tool an Add Geographic Area window will open and prompt for latitude and longitude coordinates. After supplying the necessary coordinates the same process is followed to acquire digital elevation datasets. (DOES NOT FUNCTION AT PRESENT)

DOCUMENTS Tools

There are a few Documents tools to use.

-  - Map Information and Metadata
-  - Legend
-  - Map Refresh
-  - User Instructions (Help)

When you press the *Map Information and Metadata* button an OGC Map Information window opens containing latitude and longitude coordinates in decimal form, map scale factor, and a list of information on the various available layers.

The *Legend* icon just refreshes the Layers window on the right-hand side of the page.

Map Refresh simply zooms the map out to full extent again and resets all defaults.

Clicking the *User Instructions* icon opens a help window which overviews how to navigate the seamless map, use all the tools and describes the various sections of the webpage.

Viewer Window (MAP)

The map interface is contained in window in the center of the webpage.

It is on this map that you perform functions with the selected tools.

A compass is visible in the lower left-hand corner of the map. It is a simple North Arrow indicator.

The borders of the map window also contain a "Pan" function. When the mouse arrow is directly over a border it glows red. Clicking on the border will pan the map almost an entire screen over from its former position.

An important place to be aware of is the apparent empty space below the map window. On occasion, when using tools, this space will fill with important information regarding the map.

Scale Information

The scale information box is located in the upper right-hand corner of the webpage.

"PIC"

Within this section the current scale factor of the map is displayed. For example the above image has a 1:1,454,657 factor. This specifies that every inch measured on the map interface corresponds to 1,454,657 actual inches. You may also select the scale factor and information on the scale units and calculations will appear below the map window.

It is also possible to zoom using the scale information section. This can be accomplished in two fashions:

1. The Out / In magnifying glasses.
2. The scale itself.

    Using the Out / In magnifying glasses is easy. To zoom out a scale factor of one click the Out magnifier, and the In magnifier for zooming in.

    To use the scale simply click on the scale to the left or right of the highlighted scale bar. The map will zoom accordingly. If you hold the mouse over the scale bars for a few seconds a tip window will materialize indicating the zoom factor.

Layers

The layers section is located directly below the scale information in the right-hand column of the webpage.

Layers are separated into two distinct categories:

1. Display
2. Download

    The Display tab under Layers contains all data that can be overlaid on the map. Most of all the information is separated into appropriate groups such as; Transportation, Land Cover, and Orthoimagery.

    The Download tab contains available downloadable data for the map, again organized under selective groupings.

GSDA:Obtaining a DEM from USGS

Obtaining a DEM directly from WMS

The WMS get data tool and the get data from map button allows you to download elevation data at various resolutions for anywhere in the world. The WMS developers recommend using this tool to bring elevation data into WMS. For more information about these tools and the possible data sources available using these tools, visit the Import from web [1] information page on this wiki.

DEMs from USGS web sites

[1:24K Tips]   [1:250K Tips]

The DEMs distributed by the Seamless Data Distribution System (SDDS) can be downloaded in several different formats. WMS users should choose to download data in either the Gridfloat or BIL formats, as WMS supports these formats.

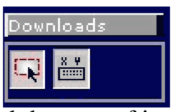
```
Click here for instructions on importing NED data into WMS
```

```
Click here to find out more about WMS.
```

The 1:250,000 scale DEMs are downloaded in the native USGS *.dem format and have a resolution of approximately 90 meters.

```
Click here for instruction on DEM data (native or SDTS) into WMS
```

Obtaining a DEM from the NED (National Elevation Dataset)

1. Go to the SDDS-Enhanced website at http://seamless.usgs.gov, or select the "DEM Data from USGS Seamless Map" link.
2. Click the appropriate titled image on the right, such as, **View and Download United States Data**.
3. Zoom in on the desired area by either single clicking or drawing a bounding box around the area you wish to view.
   - **HINT:** Dragging a box around an area will zoom in faster than clicking on a point. The Zoom In tool  is located on the left-hand side of the page under the Zoom heading and should already be selected upon opening the website.
4. Continue to zoom in on the map until the desired region is displayed.
   - **HINT:** To better recognize locations, you can turn on the display of various layers, like GNIS Cities, County Labels, and Streams. These layers are located to the right of the map window. Once selected (or de-selected) the map will automatically update.
   - **NOTE:** As you zoom in closer more layers become available for display. Some layers like GNIS Cities and Interstates are defaulted on.
5. Select the Extract Box tool 

, which is also located on the left-hand side.
6. Drag a bounding box around the area of interest.
   - **NOTE:** Upon completing the extraction box, the order will be immediately processed. After a few moments a new page will appear with a summary of your request.
7. On the SDDS Request Summary Page, click the **Modify Data Request** button.
8. Make sure the *NED 1 Arc Second* dataset is checked and select *Gridfloat* for the data format box.

- **NOTE:** If the data format box does not initialize, un-select and re-select NED quickly. A sure-fire way to fix this glitch is to select the dataset, click the Save Changes & Return to Summary button, then retry the Modify Data Request button, the data format box should be available.

9. Click the **Save Changes & Return to Summary** button.
10. Click the **Download** button located to the far right of the NED row.
11. When prompted, choose to save the zip file to disk.
12. Decompress the zip file (need help decompressing?).

The DEM has now been successfully obtained, and it is ready to be read into WMS.

```
Click here for further help navigating the USGS Seamless Map.
```

Obtaining a 1-degree (1:250,000 scale) DEM

1. Choose the Click Here button next to "to obtain 1:250K (~90m) DEM data from..."
2. Click on an area of the US map that you're interested in. You should now see a map with black squares drawn over it - each representing a 1:250,000 quadrangle.

   NOTE: Usually you'll only need to click once to zoom into a final map displaying 1:250,000 scale quadrangles from which to choose.
3. Click on the quadrangle you need.

   A list of downloadable files will now be displayed.

   NOTE: Each 1:250,000 quadrangle is normally divided into two parts: East and West. Both parts should be offered in a compressed and uncompressed format.
4. Select the format of your choice and proceed with the download by saving the file to your computer.

   HINT: It is best to get the compressed file, much shorter download time. For help on decompressing, click here.
5. Decompress the file.

   NOTE: Unlike 7.5-minute DEMs, the 1-degree DEMs are not usually in SDTS format and have only one DEM file associated with them. You have successfully obtained the DEM, but may need to download more than one DEM to completely cover your watershed. Don't forget to format (if necessary) the DEM data before importing them into WMS.

To access a list of states instead of the Graphical Map click here [1].

References

[1] http://edc.usgs.gov/glis/hyper/guide/1_dgr_demfig/states.html

GSDA:Obtaining Bathymetry Data from Geodas Design-a-Grid

GEODAS stands for **GEO**physical **DA**ta **S**ystem. It is a database system developed and managed by the NGDC (National Geophysical Data Center), a division of NOAA. GEODAS provides a tool where you can specify an oceanic, sea or lake area you wish to study and find bathymetric data for.

1. Choose the "Obtain bathymetry data from GEODAS Design-a-Grid" link, or go directly with this link http://www.ngdc.noaa.gov/mgg/gdas/gd_designagrid.html.
   This will take you to the GEODAS Design-a-Grid on the NGDC website.
2. First thing you need to do is enter a grid name in the Grid Id field.
3. Next you must choose what database you wish to get your grid information from. Select the **US Coastal Relief Model Grids** option from the *Grid Database* dropdown list.
   NOTE: Each database will have the adjustable parameters listed with their current settings described.
4. It is necessary to set a few parameters for your grid.
   1. The first parameter is the *latitude/longitude* area bounds of your grid. This is where you indicate the size and location of the study area, and thus it is essential that you specify the correct coordinates. Click the **Lat/Lon...** button. Input your coordinates into the proper bounding boxes. Once finished click **OK**.
      Need help finding coordinates, use the Coastal Relief Model webpage [1].
   2. Another important parameter is the grid format. You have to specify how you want the data to be formatted. Click the **Format...** button. On the *GEODAS: Grid Format* window choose the **XYZ (lon,lat,depth)** format option. Leave the others on their defaults. Click **OK**.
   3. Leave the other parameters on their defaults.
5. Now that the parameters are set it is possible to download the data. Click the **Design-a-Grid** button near the top of the page. You will be taken to another page confirming your grid file creation.
6. Click the **Next** button. You will be taken to another page where you can compress and retrieve your file.
7. Click the **COMPRESS AND RETRIEVE** button. You will be taken to another page that indicates the size and possible download times for your file depending on the connection.
8. Click the **Retrieve** button. A File Download window should open up, prompting you to save the file.
9. Hit the **Save** button and save it to your computer.
10. Decompress the zip file (need help decompressing?).

The bathymetry data that you have sought is now yours. It can be readily imported into WMS and SMS.

References

[1] http://www.ngdc.noaa.gov/mgg/coastal/coastal.html

GSDA:Obtaining Estuarine Bathytmetry Data from NOAA's NOS

NOS stands for **N**ational **O**cean **S**ervice. The NOS is one of five division of NOAA and is dedicated to the study, measurement and protection of oceanic environments. Bathymetry datasets for selective estuarine environments along the US coastline is sponsored by the NOS and is made available for download in *.dem format.

1. Select the link "Obtain estuarine bathymetry data from NOAA's NOS" or go directly with this link http://estuarinebathymetry.noaa.gov/finddata.html.You will be taken to the NOS Estuarine Bathymetry Find Data webpage.
2. There will be a regional map dividing the US coastlines. Choose the region you wish to explore. A page listing the estuaries with available bathymetry data will open.
   NOTE: Not all estuaries have been processed for bathymetric data, thus your preferred area might not be obtainable.
3. Select the estuary of your choice, and click the **Submit** button. Another page will open displaying the selected estuary and links for downloading data.
4. Click on the 

   button. This is the best resolution with data from 7.5 minute quads, but they have already been combined so you don't have to load numerous individual quads into WMS.
5. A *File Download* window should open up, prompting you to save the file.
6. Hit the **Save** button and save it to your computer.

You now have bathymetry data which can be easily imported to WMS and SMS. For help importing DEM files into WMS, follow this link.

GSDA:Using GIS Data Depot to Obtain DEM, DRG and DOQ Data

GIS Data Depot is a good source for data used by GIS and WMS. You will find a large selection of DEMs, DRG images, and DOQs.

NOTE: Only the DEM data (in SDTS format) can be downloaded for free. Or any downloads with the (green - normal) icon.

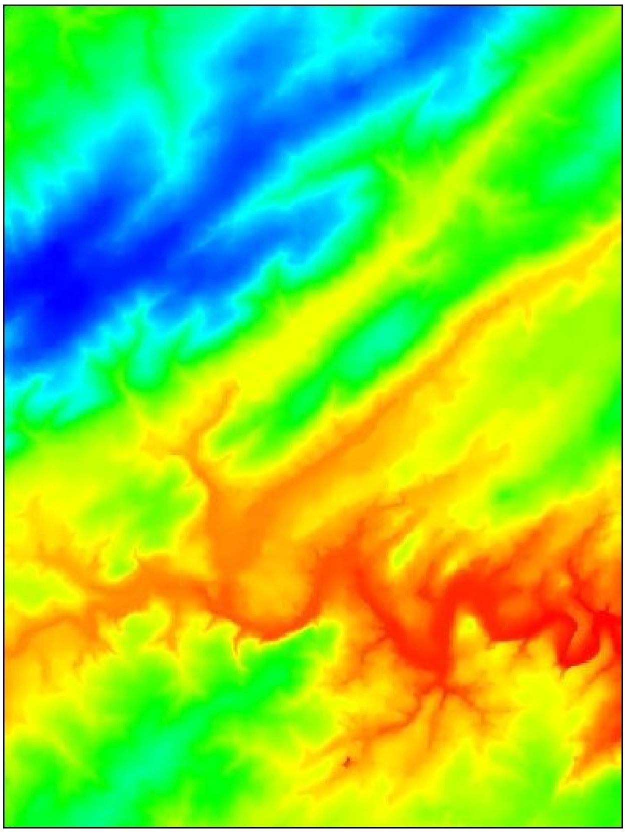
NOTE: The (gold - premium) icon indicates downloads which will cost a small fee.

Before you can pick what data to download, you must specify which country, state, and county you need data for. For the United States, the USGS Geographic Names Information System (GNIS) [1] can help you determine which county a particular feature (e.g., stream, city) is located in. To learn how to use GNIS, see Using the USGS Geographic Names Information System. The GNIS will also tell you what USGS quadrangle the feature is located in since individual DEMs and DRGs are named after their respective quadrangle.

1. Go to www.gisdatadepot.com [2].
2. Click the **Download GIS Data** button.
3. Click the state of your choice on the US map. A new window will pop open with all counties in that state with available data.
4. Click on the desired county name from the list of county links. All information accessible will be listed.
    - **TRIAL Walkthrough** (optional): To locate Deer Creek Reservoir on a trial browse-through select Utah state and Wasatch county.
5. Choose the link below which corresponds to the type of data you want.
    - How to obtain DEM data.
    - How to obtain a DRG image.
    - How to obtain a DOQ image.

Obtaining DEM Data

You can download 7.5' quadrangle DEMs (1:24,000 scale or "24K") from GIS Data Depot for free in SDTS format. Other formats may be available at a small fee. If you already know the quad name of the DEM you want continue with the steps below. If not then use the USGS Map Locator [3] to determine the quadrangle map name (see the tip on Using the USGS Map Locator). You may also use the USGS Geographic Names Information System (GNIS) [10] to find the correct quad (see the tip on Using the USGS GNIS).



Red areas are 60 meters in elevation and the blue areas are 200 meters in elevation.

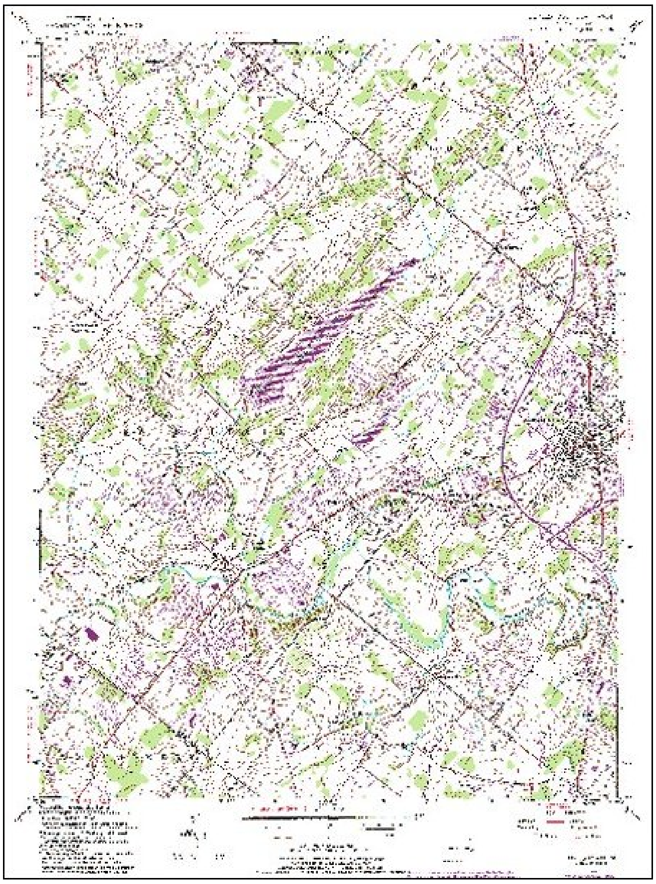
1. Click on the **Digital Elevation Models (DEM) – 24K** link.
2. Locate the link(s) for the DEM(s) you need.
   - **TRIAL WT** (cont'd): Deer Creek Reservoir is known to be in the Aspen Grove, and Charleston quads.
3. Click on the "free" Normal Download icon .
   - **NOTE:** You will be required to create an account in order to download ANY data. Money must be paid to your account before they will allow you to download any purchasable data. You will not be charged anything for normal downloads.
4. Follow the given instructions to download the file.

```
TRIAL WT (cont'd):  Aspen Grove 30 meter filename:  1694326.DEM.SDTS.TAR.GZ
                   Charleston 30 meter filename:  1635940.DEM.SDTS.TAR.GZ
                      NOTE:  Please be sure to also download and view the DEM's accompanying text file
                      for this file contains the DEM metadata.
                   Aspen Grove text filename:  1694326.DEM.SDTS.TAR.TXT
                   Charleston text filename:  1635940.DEM.SDTS.TAR.TXT
```

5. Decompress the file. See Decompressing Files for help with decompressing data files. There will be 20 files, most of which have a *.ddf extension. You have now obtained the DEM data.

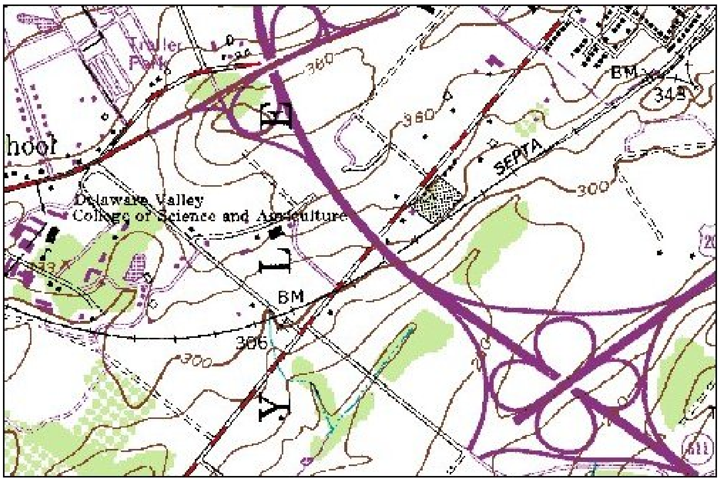
Obtaining DRG Data

Usually GIS Data Depot has the option of 24K, 100K, and 250K scale DRGs. DRGs are named after their corresponding quadrangle maps. If you need to figure out which quads you need, you can use the USGS Geographic Names Information System (GNIS) [1], (see the tip on Using the USGS GNIS). Another useful tool is the USGS Map Locator [3], (see the tip on Using the USGS Map Locator).



Doylestown, PA 24k DRG image zoomed completely out.

1. Click on the Digital Raster Graphics (DRG) - 24K link to obtain a 1:24,000 scale DRG. There could be other scale DRGs available.
2. Find which DRG you need.
   - **TRIAL WT** (cont'd): Deer Creek Reservoir is known to be in the Aspen Grove, and Charleston quads.
3. Click on the Premium Download button next to it.
   - **NOTE:** You must be logged into your GeoCommunity account in order to download. If you have sufficient download bytes remaining in your account you can download the DRGs. If you do not you will have to purchase more.
4. Follow the instructions for downloading the file.
5. Once downloaded, decompress the file.



Same DRG image as above, but zoomed in on the town.

Click here for data file decompression help. You have successfully obtained the DRG, which can now be imported into WMS. You also can view the TIFF image itself in most any graphics program such as Paint Shop Pro. The image is fairly large and will not look clear when you are zoomed out to the full extent of the image. You can zoom in to get a clearer view and more detail. This is why it is important to select the correct scale of DRG depending on how large your watershed is. Try zooming into a portion of the image such as Doylestown on the east side of the image (see below).

Obtaining DOQ Data

GIS Data Depot has limited DOQs available. There might not be any to download for your specific location at present, but they are working toward having all states covered. To locate your DOQs you will need to know the state and county it's in. Use the USGS Geographic Names Information System (GNIS) [1] to determine where it resides, (see the tip on Using the USGS GNIS). The USGS Map Locator [3] might also be helpful, (see the tip on Using the USGS Map Locator). Like DRGs, DOQs are not free at GIS Data Depot.

1. Choose the Digital Orthophotos (DOQ / DOQQ) – 1 Meter link.
2. Locate the DOQs you need and click on the Premium Download button next to it.
3. Follow the downloading instructions.

This file is in the MrSID (*.sid) format which stands for "Multi-resolution Seamless Image Database." WMS cannot directly read images in the MrSID format. However, an application is available that will save the MrSID image as a TIFF image, which can then be opened in WMS. The name of this free application is MrSID GeoViewer and can be downloaded from LizardTech.com

References

[1] http://geoname.usgs.gov/
[2] http://www.gisdatadepot.com
[3] http://store.usgs.gov/b2c_usgs/usgs/maplocator/(xcm=r3standardpitrex_prd&layout=6_1_61_48&uiarea=2&ctype=catalogEntry&carea=%24ROOT)/.do/

GSDA:Using the USGS Geographic Names Information System (GNIS)

Since all 1:24,000 scale DEMs and DRGs are named based upon the 7.5-minute quadrangle they are located in, it is important to know the proper quadrangle map names.

The USGS GNIS Query is another tool to find out which state, county, and USGS quadrangle a particular feature is located in. Feature types included in the GNIS Query are: reservoirs, streams, lakes, valleys, basins, dams, populated places (representing cities and town,) etc. Information for these features, such as the geographical coordinates (latitude / longitude,) elevation, and the USGS quad name(s) covering the feature are also listed for you.

An alternative, and possibly a better option to using the USGS GNIS page, is to use Google Maps [11] or the Microsoft Bing Virtual Earth Map Locator tool (included with WMS) to search for a location of interest. Elevation data, imagery, and possibly other data for most locations in the world can be downloaded after finding the location using the Virtual Earth Map Locator.

---

1. Connect to the internet and go to geonames.usgs.gov. [10]. You are now on the Geography Information site for USGS.
2. Click on an option to search domestic (US) names or foreign names. This will take you to the GNIS homepage.

On the feature summary page (the final page) there are often links to viewable maps of the site.

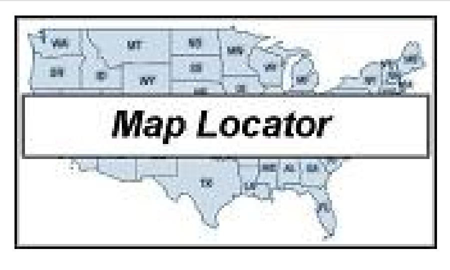
- Topozone - Opens a topographical map of the feature.
- TerraServer - Views the following images on the MSN TerraServer.
  - DRG - Digital Raster Graphic (digitized topographic map.)
  - DOQ - Digital Orthophoto Quadrangle (black and white aerial photograph.)
- USGS map - Generic map window using US Census Bureau's Tiger Map Server.
- EPA - Goes to the corresponding watershed profile on the EPA "Surf Your Watershed" site.

USEFUL LINKS

[USGS GNIS Home [10]]]

Feature Name:	**Deer Creek Reservoir**
Feature Type:	reservoir
Elevation:	5417
State:	Utah
County:	Wasatch
Latitude Longitude	**USGS 7.5' x 7.5' Map**
402425N 1113139W	Aspen Grove
402705N 1112911W	Charleston

GSDA:Using the USGS Map Locator to Find 7.5' Quadrangles



All 1:24,000 scale DEMs and DRGs are named after the 7.5-minute quadrangle they are located in. Therefore it is essential to know the correct quad names for your area of interest.

There is a useful tool for finding these map names. It is the Map Locator inside the USGS Store on the USGS website. This particular Map Locator can be used to determine the proper 7.5' x 7.5' quadrangles (as well as 7.5' x 15' quadrangles) for the area you choose by means of an interactive map.

1. Connect to the internet and go to www.usgs.gov [3].
2. Click on the **Publications** tab at the top center of the page. This will take you to the publications and products page.
3. The **USGS Store** should be listed first, click it.

   NOTE: You do not need to order anything or login and give personal information to use the Map Locator.
4. Next click the **Enter USGS Store** link.

   NOTE: Now you are in a session on the store webpage. From this point you may use two pathways in discovering the quadrangle maps of your choice. Here we will follow the Map Locator link. The other pathway is a link to the GNIS Query which is directly beneath the Map Locator. To get help utilizing the GNIS Query go to the link at the bottom section of this webpage.
5. Click the **Map Locator** to enter the mapping tool webpage. It will look like the following:

   "PICTURE"
6. Specify the area of your choice in the **Select State** or Region dropdown list. For this example use Utah. Clicking on the map window will also zoom in on the location you desire.

   HINT: Using the *Select State or Region* dropdown list will zoom in quicker than the alternative method.
7. Continue zooming in by clicking on the **Salt Lake City** dot.

   NOTE: Greater populated cities will show as red dots on the map.

   "PICTURE"
8. Continue zooming in by clicking the **Provo** dot. Now the quadrangle map names appear in green on the map window.

   "PICTURE"

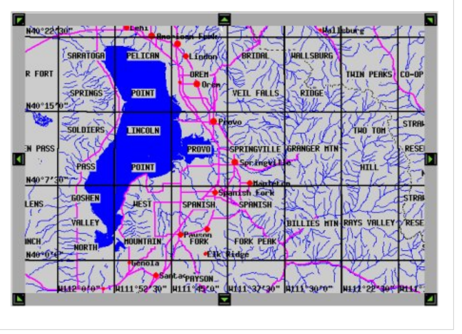
   The maps viewable in the window will also appear listed beneath it with their respective names and scale information.

   HINT: If additional maps are needed it is effective to use the pan arrows on the borders of the map window.

You now have the proper names for 7.5' x 7.5' quadrangle maps. This information is useful when looking for DEMs and DRGs since they usually are named after their respective quads (at least 7.5' quadrangles.)

USEFUL LINKS

[USGS Home [3]] [USGS Store [1]] [Map Locator [2]] [GNIS Query Help [3]]



References

[1] http://store.usgs.gov/
[2] http://store.usgs.gov/cgi-bin/map_locator.pl
[3] http://emrl.byu.edu/gsda/data_tips/tip_using_gnis.html

7.2.c.2. Hydrography Data

GSDA:Hydrography Data

Stream Depths, Flow Rates, and Forecasts

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [1] Obtain stream stage data from the USGS WaterWatch [2]

Water Watch offers real-time stage and discharge data for about 3,000 stream gages. Plus, it is linked to NWIS-Web, which furnishes surface and ground water data, as well as water quality data.

Advantages:

> Easy-to-use graphical map allows users to access data for a gaging station in a couple of clicks. Users can quickly access a list of all stations within a state.

- Help - Obtaining stream-stage data from WaterWatch

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain stream stage data from the USGS NWISWeb [10]

NWISWeb offers an extensive amount of data, including surface and ground water data, as well as water quality data.

Advantages:

> Users can search for a gaging station by its name, ID number, State, Drainage Basin, and more. Use this site if you need advanced search capabilities.

- Help - Obtaining stream-stage data from NWISWeb

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png [3] Obtain stream forecast data from the USDA/NRCS [4]

The USDA/NRCS offers stream forecast data as well as current water supply maps.

- Help - Obtaining stream forecast data from the USDA/NRCS

Rivers, Lakes, and Seas

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_webGIS.png [5] Obtain hydrographic data from WebGIS [5]

WebGIS provides data in shapefile and standard DLG formats

- Help - getting hydrographic data from WebGIS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_ESRI.png [1] Obtain hydrographic data from ESRI Census 2000 TIGER [5]

ESRI and the U.S. Census Bureau offer hydrographic data in shapefile format for ArcView or ARC/INFO

- Help - Obtaining hydrographic data from ESRI Census TIGER 2000

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain hydrographic data from the USGS/EPA [6]

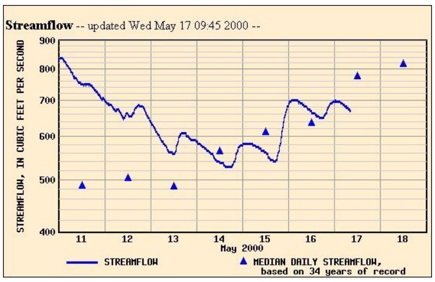
The USGS/EPA offers hydrographic data for each CU (cataloging unit) in the US in either ESRI/ARC or USGS/SDTS format.

- Help - Obtaining hydrographic data from the USGS/EPA

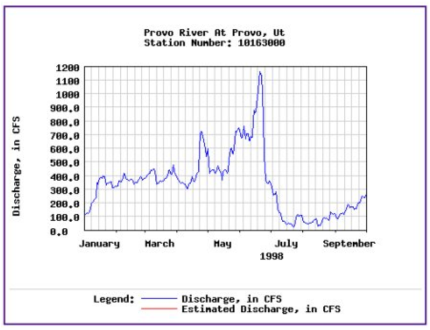
Stream Data Overview

Stream stage simply refers to the water depth in a stream or river. The water depth is related to the flowrate and rating curves (plots of stage vs. flowrate) have been created for many streams and rivers. Available data sources can give real-time (i.e., current to the moment) stream-stage data whereas other sources give historical figures along with statistical data such as median flowrates etc. Some examples are given below for the Provo River in Utah. Another important flow data type are flow-duration curves (also called reliability curves) which give engineers an idea of what flowrates can be expected a certain percentage of the time. An annual hydrograph is required to create a flow-duration curve which is shown in the third figure. Streamflow data used for forecasting is also available for specific streams or for regions in general.

Graph showing stream stage (depth) vs. time.



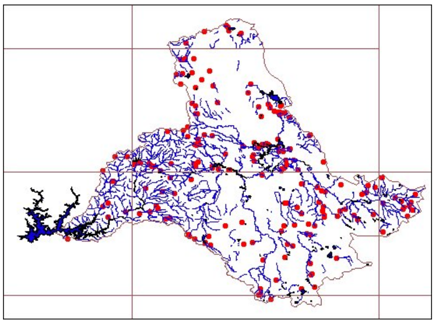
Graph showing flowrate vs. time (a hydrograph) for the same river in the above figure.



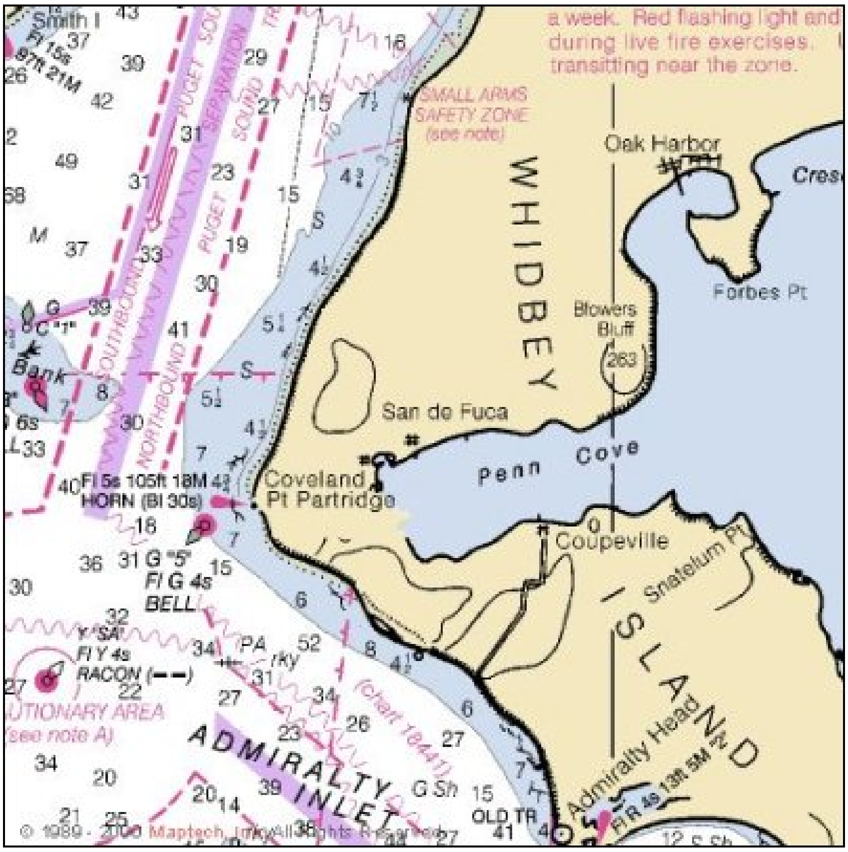
An annual hydrograph for the Provo River for station # 10163000.

Hydrographic Waterbody Overview

By definition, hydrography is the study and survey of rivers, streams, creeks, springs, wells, ponds, lakes, reservoirs, oceans, seas, bays and estuaries with respect to their tides, flow characteristics, and navigability. Much of the available hydrographic information is in a format compatible with a GIS such as ArcView. Locating streams and other water bodies may be essential in creating a good model.



USGS/EPA NHD (National Hydrography Dataset) showing stream networks and lakes.



Stream Data Tips

- Help - Obtain stream stage data from USGS NWISWeb?
- Help - Obtain real-time stream stage data from USGS WaterWatch?
- Help - Obtain stream forecast data from the USDA/NRCS?

Hydrographic Waterbody Tips

- How do I obtain hydrographic data from WebGIS?
- How do I obtain hydrographic data from the USGS/EPA?
- How do I obtain hydrographic data from ESRI Census TIGER 1995?
- How do I decompress data files?
- How do I import hydrographic data into WMS?
- How do I import a standard DLG file into WMS?

References

[1] http://www.usgs.gov/waterwatch/
[2] http://water.usgs.gov/waterwatch/
[3] http://http://www.nrcs.usda.gov/
[4] http://www.wcc.nrcs.usda.gov/wsf/
[5] http://www.esri.com/tiger
[6] http://nhdgeo.usgs.gov/viewer.htm

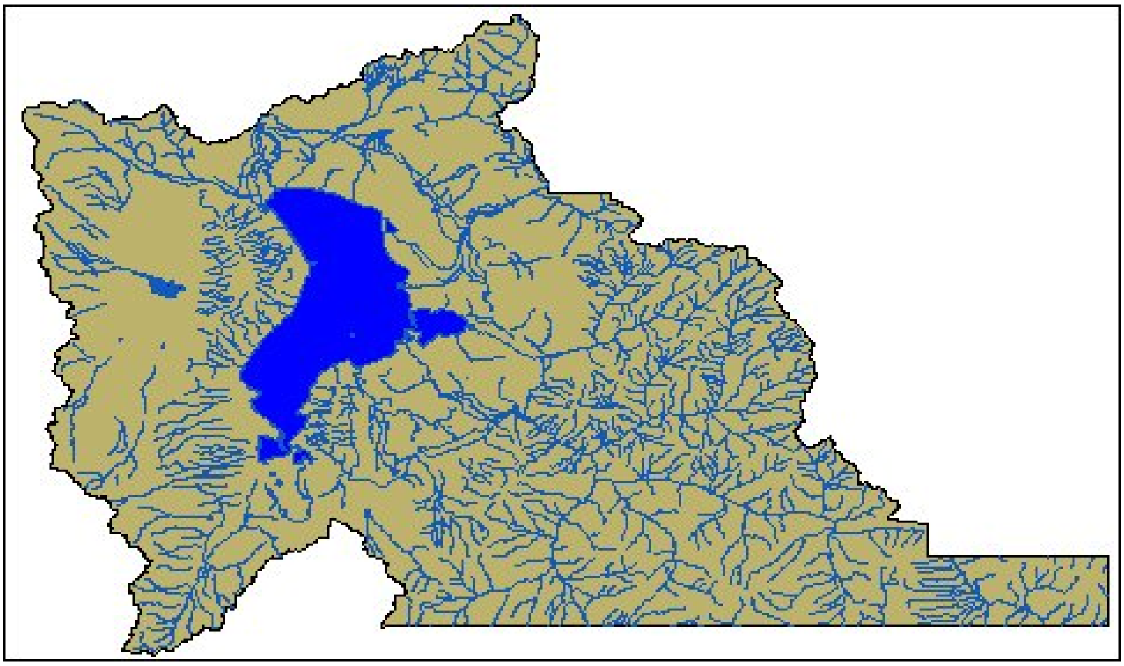
GSDA:Obtaining Hydrographic Data from ESRI Census TIGER 1995

To obtain hydrographic data from ESRI, go to their Census TIGER 1995 web site at www.esri.com/data/online/tiger/index.html [1]. If you have trouble using this link, follow the bulletted steps below:

- Go to the ESRI home page at www.esri.com [2].
- Choose the *DATA* button near the top.
- Click the *ArcData Online* link.
- Choose the *Browse ArcData Online* button.
- Choose the *DATA PROVIDER* button.
- Scroll through the list of providers and choose the *Census TIGER Data* link.

1. If you have **NOT** registered, click the *Register to Download Data Now!* link to register before you continue.
2. If you **HAVE** registered, simply enter in your email address which you used to register and choose the *Enter* button.
3. Select a state from the list or from the map. For example, choose *Utah*.
4. You may now select either a *subregion* (e.g., county) or a *data layer type*. For example, choose the data layer type described as *Line Features, Streams* and click the *Submit Selection* button.
5. Either choose a county from the list or choose to download all counties. For example choose Utah county only.
6. Click on the *GENERATE FILE NOW* button followed by the *DOWNLOAD DATA NOW* button.
7. Proceed to download, save, and uncompress the data. Click here for more help on uncompressing files. Once the data file is uncompressed, you will see *another* .zip file which you *also* need to uncompress. You've now successfully obtained hydrographic data from ESRI. In order to obtain data on lakes, the same steps could be followed as above, but instead of choosing the *Line Features, Streams*, select the *Land and Water Outlines*

option. Be sure to format (if necessary) the data before importing them into WMS.



References

[1] http://www.esri.com/data/online/tiger/index.html
[2] http://www.esri.com

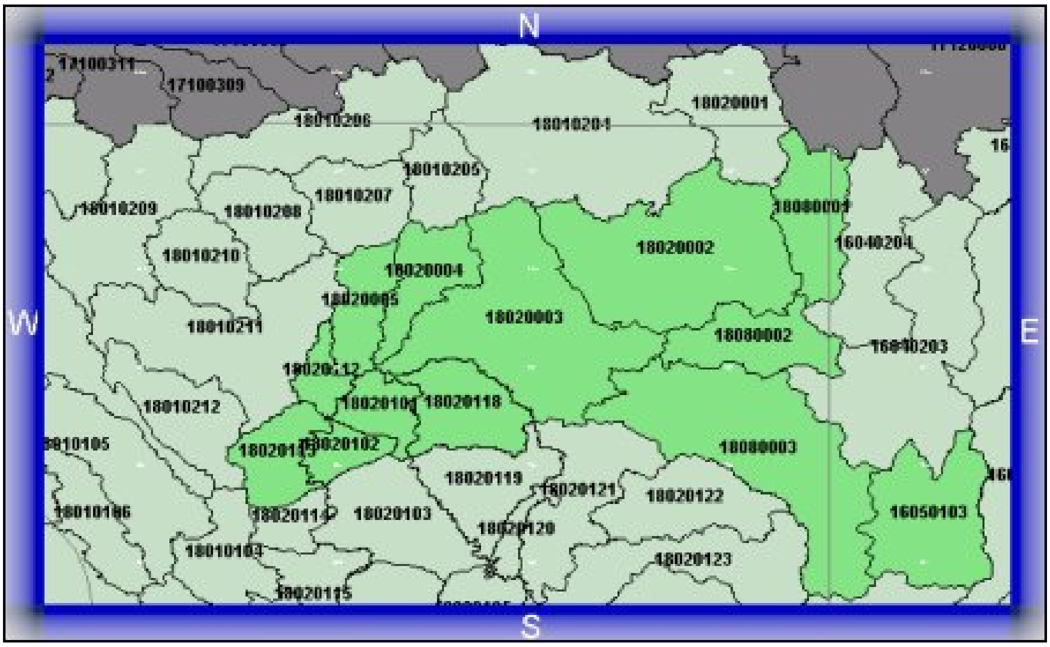
GSDA:Obtaining Hydrographic Data from the USGS / Epa

The National Hydrography Dataset (NHD) offers Arc/Info workspaces of hydrologic Cataloging Units and their accompanying data. The NHD Interactive Map website can be found at http://edcnts14.cr.usgs.gov/Website/nhdserver/Run.htm [1]. If this link does not work, follow the bulletted steps below:

- Go to the USGS home page at www.usgs.gov [2].
- Choose the Water button near the top.
- Under the "Water Data" heading, click on the GIS link.
- Click on the NATIONAL HYDROGRAPHY dataset link.
- Choose the Data button at the left.
- Select the Obtaining NHD Data link.

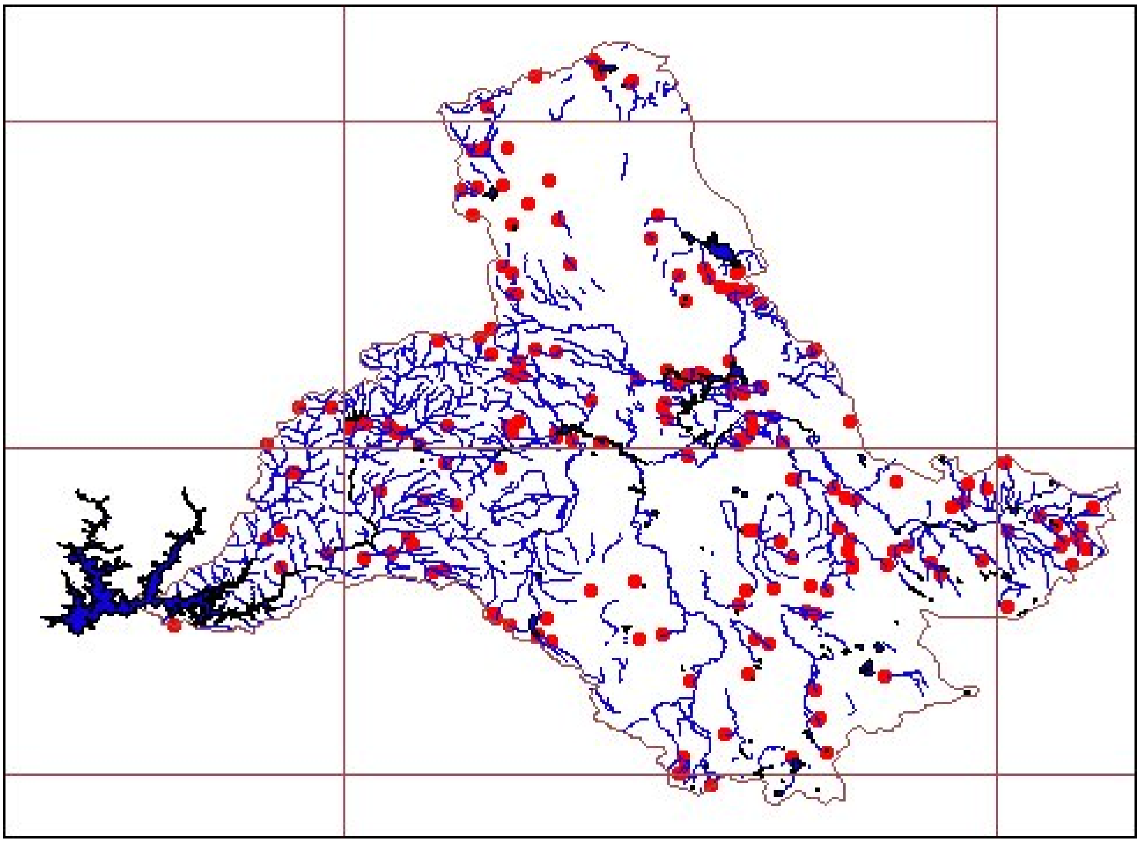
You will now see a map of the United States and Puerto Rico. To the left of the map window are the tools to help you in navigating and selecting your data. Below the map window is a text string that displays the current, or active tool.

The United States is divided into cataloging units (CU). You can interactively click on the map to zoom to a region of interest. You can also control the zoom factor (e.g., 2x, 4x). The CUs are classified as (1) CU Distributable, (2) CU Distributable * Initial Release, and (3) CU in Progress. You will not be able to download a CU in Progress. As you zoom in, the individual CUs with their number are displayed (see image below).



Zoom in on the map so that you can distinguish individual CUs. At this point you can now do two things:

- Check the CU Identify option and then select a CU with your mouse to view the metadata (i.e., data about the data) for that specific CU.
- Check the CU Download option to begin downloading the hydrographic data for that specific CU.
  1. For example, check the CU Download option and click on any CU.
  2. You will be required to enter your name, but all other information is optional. A pre-download screen will appear and may take a few minutes to begin the download process.
  3. If you get a security window, click on the Grant button to continue.
  4. You will now get the choice to download ESRI/ARC or USGS/STDS data.
  5. For example, choose the ESRI/ARC option and click the Download button to begin the download process.
  6. After the CU has been saved, uncompress it with a utility such as WinZip. See Decompressing Files for help on uncompressing files. The NHD data can then be brought into a GIS such as ArcView (see image below).



References

[1] http://edcnts14.cr.usgs.gov/Website/nhdserver/Run.htm
[2] http://www.usgs.gov

GSDA:Obtaining Hydrographic Data from WebGIS

To download hydrographic data in shapefile format from WebGIS:

1. Go to the webGIS home page at www.webgis.com [1]
2. Click on the Shapefile (UTM) or Shapefile (Lat/Long) link under the Digital Line section towards the bottom
3. Click on the state of choice
4. Click on the county of choice

   Note that as your mouse icon moves over a county boundary, its name is displayed in the browser's status bar. Alternatively, you can list the counties alphabetically.
5. Click on the location(s) you wish to download
6. Save the zip file to your computer
7. Decompress the zip file

   Help – Decompressing Files

The shapefile can now be imported into WMS

- Help – Importing hydrographic shapefiles into WMS

To download hydrographic data in standard DLG format

1. Click on the Standard (UTM) link under the Digital Line section toward the bottom
2. Choose your state and county as indicated above
3. Click on the area you wish to download
4. Save the zip file to your computer
5. Decompress the zip file

   Help – Decompressing Files

The DLG data is in a single file with an *.opt extension and can be imported into WMS

- Need help importing this DLG file into WMS

References

[1] http://www.webgis.com

GSDA:Obtaining Stream Data from the USGS WaterWatch

Water Watch allows users to compare stream flow data nationwide, and to quickly access stream data for individual gage stations. The web site's interface permits choosing between real-time data (the default), recent daily data, an assortment of historical surface water data, and more. Refer to the following table for some brief details on the three types of data:

	Real-Time	Recent (Daily)	Streamflow (historical)
Availability Period	31 days prior to latest recording	18 months prior to latest recording	Period of Record for each site
Recording Increment	5 to 60 minutes (mostly 15 min.)	daily average	daily average

Steps for obtaining the stream-flow data:

1. Go to the Water Watch [2] website.
2. Click on the state of choice.
3. Click on a gage station point (notice that if you hold the mouse pointer over a station, information pops up regarding that station, including the station name in all-caps).

   Alternatively, you can click the "List all stations" option under the state map in order to select the from a list of all the gaging stations in the state, grouped by river basin name.

   Hint: When viewing the list of all stations in a state, you can quickly search for a station name by using your browser's Find tool (select Find on this page... from the Edit menu).
4. Scroll down the page and examine the discharge data and any other available data, such as gage height (at some gages, precipitation data is also available).

   Note: You can change the way the discharge data is displayed and the number of days of data by choosing different values in the cream-colored table at the top of the page.

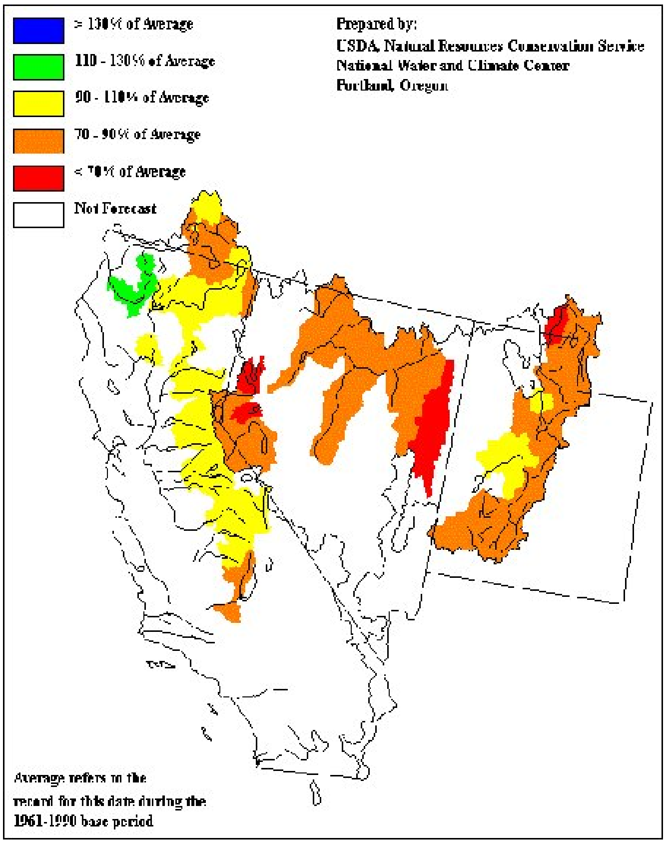
   Output Formats available: Graph, Table, Fixed-Width Table, and Tab Separated.

GSDA:Obtaining Stream Stage Data from the USDA/NRCS

Go to the USDA/NRCS National Water & Climate Center water supply site at www.wcc.nrcs.usda.gov/water/w_qnty.html [1]. If you have trouble with this link, follow the bulletted instructions below:

- Go to the USDA home page at www.usda.gov [2].
- Choose the Agencies, Services & Programs button near the top.
- Click on the Natural Resources Conservation Service (NRCS) link near the bottom.
- Choose the Technical Resources button at the left.
- Click on the Water and Climate Data link near the bottom.

1. Choose the Water Supply Forecasts link.
2. Click on the Spring & Summer Streamflow Forecast Maps link.
3. For this example, click the Great Basin and California link.
4. Choose the April 2000 button. The image should look similar to the one below.



1. Using your browser, backtrack to the Water Supply Forecasts page at www.wcc.nrcs.usda.gov/water/w_qnty.html [1].
2. Click the Streamflow Forecast Probability Charts link.
3. Choose Utah as the state, Utah Lake, Jordan River & Tooele Valley as the basin, and April 2000 as the month/year.
4. Choose the Submit button. Several streams, rivers and creeks will be displayed along side bar charts. These charts show the percent probability that the flowrate will exceed the average by a specified percent (see image below). For example, there is a 45% chance that PAYSON CK will exceed the average flowrate by 70% (or 4.4 times 1.7).

[PICTURE GSDAImage074.bmp]

Data for reservoir storage are also available at this web site.

References

[1] http://www.wcc.nrcs.usda.gov/water/w_qnty.html
[2] http://www.usda.gov

GSDA:Obtaining Stream Stage Data from the USGS NWISWeb

NWISWeb provides historical, recent, and real-time data. The real-time portion of the site is very closely related to the USGS WaterWatch website, where users can click on a US map begin the data retrieval process.

NWISWeb offers an immense quantity of data. Streamflow data is offered in three categories: Real-Time, Recent (Daily), and Streamflow (daily data for the site's period of record, termed "historical" in this page). This page shows how to obtain historical Streamflow data from the NWIS. Take a look at the following table to get a idea of some of the basic differences in each data catergory:

	Real-Time	Recent (Daily)	Streamflow (historical)
Availability Period	**31 days prior to latest recording**	**18 months prior to latest recording**	**Period of Record for each site**
Recording Increment	**5 to 60 minutes (mostly 15 min.)**	**daily average**	**daily average**

To obtain data, go to the USGS NWISWeb site at http://waterdata.usgs.gov/nwis. If you have trouble getting there, follow the bulleted steps below:

- Go to the USGS home page at www.usgs.gov [2].
- Click on the *Water* button near the top.
- Click on the *NWISWeb* link located in the green table on the right-hand side of the page.

The following instructions describe how to obtain historical streamflow data from NWISWeb:

1. Click the Surface Water button.
2. Click the Streamflow button.
3. Place a check in the boxes of the site selection criteria you wish to use.

   If you don't know many details about the site, try a search based on State and Site Name.
4. Click the Submit button.
5. Under the Select Sites heading, enter your site selection criteria.
6. Under the Choose Output Format heading, select the "Table of sites sorted by" option (this is the default).
7. Select "Site Name" from the *sorted by:* list and "County" from the *grouped by:* list.

   These options were chosen for this illustration, but the other options can be useful as well.
8. Click the Submit button at the page's bottom.
9. Select a link for the site of interest.
10. Enter the desired time interval in the retrieve data from: fields.

    Note: Refer to the Period of Record dates in selecting the time interval.
11. Choose an output option, either *Graphs of data* or *Tab-Separated Data*.

    If you choose to display a graph, the data will be displayed graphically in the browser;

    If you choose to retrieve tab-separated data, you can further choose to save the data to a file, save it to a compressed file, or display it in the browser.

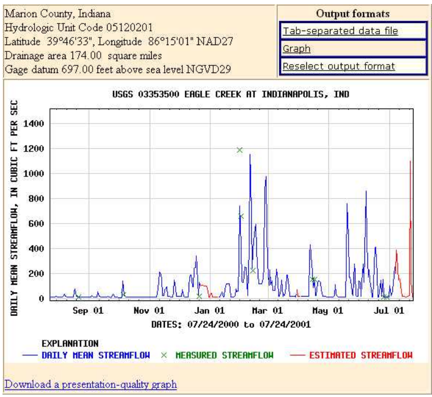


Figure 1: A Sample Daily Streamflow Graph from USGS' NWISWeb.

7.2.c.3. Imagery

GSDA:Imagery

DRG Image Data

Obtain images directly from WMS using the Get Online Maps tool [1]

WMS offers an option to download various types of high-quality image data for anywhere in the world using the Get Online Maps tool. This is the recommended method to get image data into WMS unless you have other image sources that may be more recent or better quality.

Obtain a DRG from MSR Maps [2]

MSR Maps offers USGS Topos in several different resolutions. WMS uses this web service to download imagery for the United States.

Obtain a DRG from other public sites [3]

Sometimes, the best way to locate GIS data from a public source, such as a state or county agency, is to consult an internet search engine. Go to a search engine website, such as www.google.com [4] and enter the search criteria, such as "DRG Georgia."

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_GeoCommunity.png [5] Obtain a DRG from GIS Data Depot [6]

- The USGS GNIS [10] and Google Maps [11] are useful tools for locating the correct DRG.
- Need additional help using USGS Map Locator?
- Need additional help using the USGS Geographic Names Information System (GNIS)?
- Need additional help obtaining a DRG from GIS Data Depot?

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png [12] Obtain a DRG from Land Info [12]

Land Info is a commercial site offering a wide variety of spatial data for purchase.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain a DRG from the USGS [7]

You may purchase DRGs from the USGS.

DOQ Image Data

Obtain images directly from WMS using the Get Online Maps tool [1]

WMS offers an option to download various types of high-quality image data for anywhere in the world using the Get Online Maps tool. This is the recommended method to get image data into WMS unless you have other image sources that may be more recent or better quality.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain a High Resolution DOQ from The National Map [8]

This USGS site offers 1/3 meter resolution Orthos for various metropolitan areas.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_GeoCommunity.png [5] Obtain a DOQ from GIS Data Depot [6]

- The USGS GNIS [9] can be a useful tool to determine the *name* of the correct DOQ (GIS Data Depot also requires that you know the county).

- Additional help obtaining a DOQ from GIS Data Depot
- Additional help using the USGS GNIS (Geographic Names Information System)

Obtain a DOQ from other public sites [3]

This list maintained by the USGS offers DOQ data for a selection of states.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_TerraServer.png [3] Obtain a DOQ from Terraserver [3]

- Additional help obtaining a DOQ from Terraserver
- Additional help uncompressing data files

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain a DOQ from the USGS [10]

You may purchase DOQs from the USGS.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png [12] Obtain a DOQ from Land Info [12]

Land Info is a commercial site offering a wide variety of spatial data for purchase.

Satellite Data

http:// / wikis. aquaveo. com/ xms/ FTPImages/ GSDA/ GSDA_USGS. png [3] Obtain satellite images from the USGS SRTM [8]

The SRTM offers 30 meter elevation data in GeoTIFF and ArcGRID formats

- [[GSDA:|Additional help obtaining satellite images from the USGS SRTM database]]

http:// / wikis. aquaveo. com/ xms/ FTPImages/ GSDA/ GSDA_TerraServerUSA. png [11] Obtain satellite images from Terraserver [11]

Among other image types, Terraserver offers OrbView and SPIN-2 satellite images for purchase.

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_LandInfo.png [12] Obtain satellite images from Land Info [12]

Land Info is a commercial site offering a wide variety of spatial data for purchase.

Nautical Charts

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png [13]

- Obtain nautical charts from NOAA. [12]
- Obtain historical nautical charts from NOAA. [13]
- Obtain additional information from NOAA. [14]

Image Overview

The use of maps and photographs within WMS/GIS serves many purposes. These images can be used as a backdrop to create a conceptual hydrologic model. In other words, the engineer can create feature objects (i.e., streams, outlets) on top of the map image to ensure that they are in the proper locations. There are three types of images generally used with WMS: TIFF, DRG, and DOQ. DRG images are the most commonly used.

In order for images to be useful they must be georeferenced. This means that the map image itself has been fit to actual coordinates on the earth's surface to some coordinate system such as UTM (Universal Transverse Mercator). If a map or image is not available for the area of interest, then any map or image can be scanned using a typical scanner. This image can then be georeferenced within WMS.

- Learn about TIFF images and DRGs.

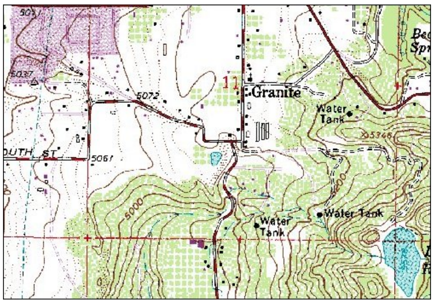
- Learn about DOQ images.
- Learn about satellite images.

A common type of image format is a TIFF (tagged image file format) image. If a TIFF image is not already available from some source, they can be easily created by scanning a map using a typical scanner.

A special type of TIFF images that is already georeferenced is called a GEOTIFF, and does not need to be georeferenced with any additional steps since the georeferencing information is contained within the image itself. Some software will read GEOTIFF images and some won't. Other TIFF images come with a separate file called a TIFF World File which contains the georeferencing information. Finally, if a TIFF image is not georeferenced for some reason, it can be done in WMS or GIS through a process called registration (see WMS tutorial for more information).

DRGs are a special type of TIFF image. The USGS has scanned their standard series topographic maps and created georeferenced TIFF images from them - images like these are known as a DRG (Digital Raster Graphic). They are usually available in 1:24,000 (7.5' quads), 1:100,000, and 1:250,000 scale. The scale required for a project in WMS is very important since DRGs are not normally tiled together as are DEMs, but the proper size should be selected for the project (i.e., a very large watershed may require a 1:250,000 scale DRG). DRG images are often large with respect to disk space, often exceeding 10 megabytes each. Most DRGs are GEOTIFFs.

WMS and GIS will both read a DRG, and most graphics programs will at least display one on the computer screen. The USGS offers free software that will also read a DRG image (visit http://mcmcweb.er.usgs.gov/viewers/dlg_view.html for more information).



A portion of the Draper, Utah 1:24,000 scale (7.5' quad) DRG map.

DOQ images are high-resolution (often 1 square meter per pixel) computer generated images of aerial photographs shot at about 20,000 ft altitude. They are usually very large in file size (often exceeding 40 megabytes for black & white) so typically 4 DOQ images are taken for each 7.5' quadrangle as a NW, NE, SW, and SE quarter-quad. The "ortho" refers to the fact that the image displacements caused by camera tilt and terrain topography have been removed from the aerial photo (this process is called orthorectification). This means that the image is a photographic map which can be used to accurately measure distances directly from the photograph and other cartographic (map) information can be directly overlaid onto the DOQ. DOQ images are sometimes georeferenced, and sometimes they are not. DOQs come in various file formats. Some images are in .jpg format and have a file associated with them similar to a TIFF world file (as is the case with DRGs). Another common format is .bil which stands for "band interleaved by line" multiband image. ArcView and ARC/INFO require the JPEG extension to view .jpg images but will automatically read the .bil format as an image data source (versus a feature data source such as a shapefile). WMS cannot read .bil format directly. DOQ images should be available for the conterminous United States by 2004, and then will be updated every 5 or 10 years depending on how rapid the land use change is in a certain area.

A portion of a DOQ showing a wharf and boat dock.

- FAQ about DOQs. [15]
- Learn more about DOQ images at the USGS. [16]

Other image types such as SPIN-2 and OrbView satellite photographs can be found, viewed, and purchased from several sources. As mentioned in the overview, satellite images are sometimes clearer and more resolute than aerial photographs.

Image Tips

- How do I obtain image data from Terraserver?
- How do I obtain a DRG from GIS Data Depot?
- How do I obtain a DOQ from USGS?
- How do I obtain a DOQ from GIS Data Depot?
- How do I obtain a DOQ from Terraserver?
- [[GSDA:|How do I obtain satellite images from USGS?]]
- How do I use the USGS Map Locator?
- How do I use the USGS GNIS (Geographic Names Information System)?
- How do I decompress data files?
- How do I register an image for use in WMS?
- How do I import a DRG into WMS?
- How do I use WMS to convert DRG data into a different coordinate system?

References

[1] http://www.xmswiki.com/xms/Get_Online_Maps
[2] http://msrmaps.com/
[3] http://libraries.mit.edu/gis/data/datalinks/statedataweb.html
[4] http://www.google.com
[5] http://www.geocomm.com/
[6] http://data.geocomm.com/catalog/index.html
[7] http://mcmcweb.er.usgs.gov/drg/
[8] http://seamless.usgs.gov/
[9] http://mapping.usgs.gov/www/gnis/
[10] http://edc.usgs.gov/webglis
[11] http://terraserver-usa.com/
[12] http://nauticalcharts.noaa.gov/mcd/Raster/
[13] http://historicalcharts.noaa.gov/historicals/historical_zoom.asp
[14] http://www.csc.noaa.gov/crs
[15] http://mapping.usgs.gov/digitalbackyard/faqsnew.html#1
[16] http://www-wmc.wr.usgs.gov/doq/

GSDA:Obtaining DOQ Image Data from MSN TerraServer

Terraserver provides DOQs for viewing, purchasing, and sometimes free downloading. Although some of Terraserver's images are geopositioned (i.e., north = north), they are usually not georeferenced but can be for an extra fee (contact Terraserver for details).

1. Go to Terraserver at www.terraserver.com [1].
2. Click on the *View Images* link near the center of the screen.
3. Although you could use the coverage map, click on the *advanced find* button for this example.
4. Enter as much information as you know about a particular place. For example, type in "Doylestown" in the *City or Place Name* field. Type in "Pennsylvania" in the *State or Province* field. Type in "USA" in the *Country* field and choose "City" from the *Place Type* field.
5. Click the "go!" button. You could enter in more or less information than we just did, but either way Terraserver will search for your information based on what you enter—the more information you have the better. Terraserver will find any documents (i.e., maps, photographs) that match your search criteria.
6. For this example, click on the *USGS Aerial Photograph 28 Sep 1996* link under *Doylestown, Pennsylvania, United States*. You are now looking at the aerial photograph. You can zoom out, zoom in, and even download the image if you wish. While at the Terraserver site, you can click on the *Image Information* button to view latitude/longitude as well as other information about the photograph.

References

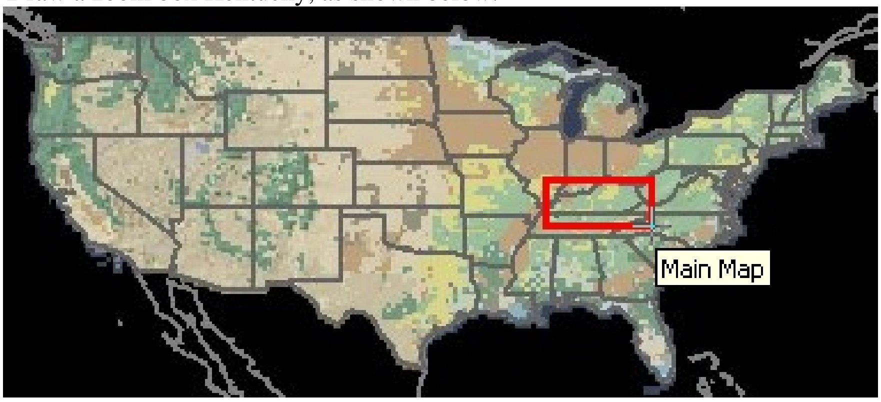
[1] http://www.terraserver.com

GSDA:Obtaining DOQs from The National Map on SDDS

The USGS has a goal to provide high resolution (one pixel = 1/3 meter) ortho photos for most of the metropolitan areas of the U.S. Currently the list of available areas includes:

- Chicago, Illinois
- Baton Rouge, Louisiana
- El Paso, Texas
- Lexington, Kentucky
- Lincoln, Nebraska
- Mobile, Alabama
- Modesto, CA
- Seattle, WA
- Shreveport, Louisiana
- St. Louis, Missouri-Illinois
- Stockton, California
- Washington D.C.

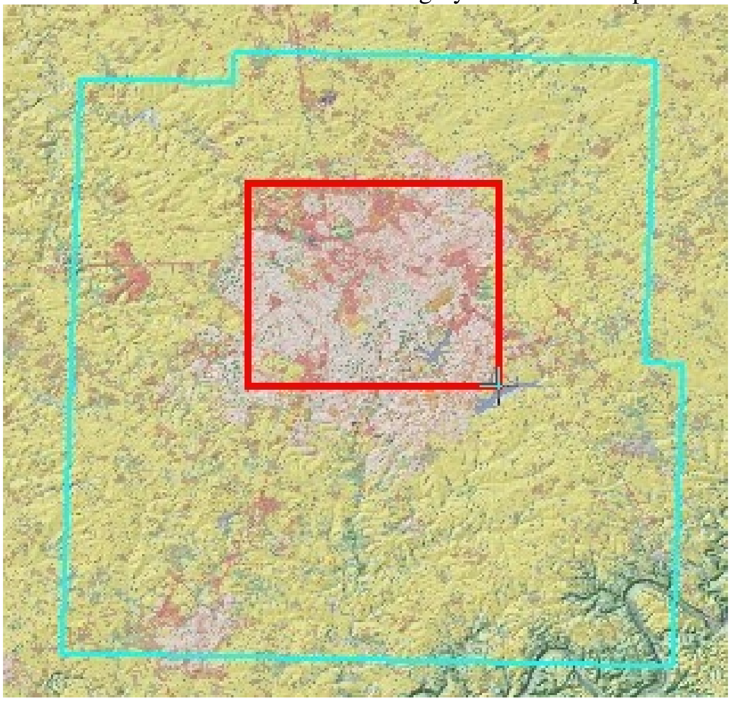
The following steps will guide you in downloading an image for Lexington Kentucky

1. Go to *The National Map* website at seamless.usgs.gov [1].
2. Click on the link labeled *View and Order Data Sets - United States Viewer*
3. Draw a zoom box Kentucky, as shown below:


4. On the right-hand side of the page, toggle on the check box for displaying the *High Res Ortho Index*


5. Scroll down and select the *Refresh Map* button
6. Draw a zoom box around the light blue area that appears on the screen

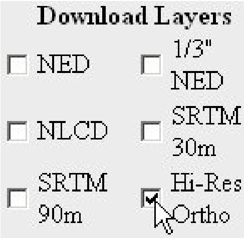
7. Zoom in on the area that is located roughly 1/3 from the top and in the middle of the blue area



8. In the Visible Layers section, toggle on the check box to display *High Res Ortho: Lexingtion, KY* (you must scroll down a bit to see it)
9. Select the *Refresh Map* button
10. Zoom in around the race track

11. Zoom in on the buildings located to the west of the track

12. In the Download layers sections, toggle on the check box to download *High Res Ortho* data and toggle off the boxes for NED and NLCD



13. Choose the *Select with Rectangle* tool located under the Download section. 

14. Draw a selection box around the buildings

15. When the order is finished being processed, select the Download link for the Ortho image
16. Save the file to your computer

The image is based in the UTM NAD 83 coordinate system with horizontal units in meters. Before you can view the image in an application, you need to decompress the downloaded file.

- Tips – decompressing files.

References

[1] http://seamless.usgs.gov

GSDA:Obtaining Image Data from MSN TerraServer

The new TerraServer USA site offers USGS Topographic maps as well as aerial photos in the JPEG format for the United States, with limited coverage of Canada, Puerto Rico, and Mexico. Along with the topo maps and aerial photos, TerraServer provides the GIS world file, which aids in geo-referencing the image in some software.

First of all you must go to the TerraServer USA site at http://terraserver.homeadvisor.msn.com/.Or use the MSN TerraServer button or link under the MSN TerraServer section.

Once at MSN TerraServer there are three main methods to choose from:

1. Navigating the map.
2. TerraServer search panel.
3. Advanced Find search.

Navigating the map on TerraServer USA

Navigating the map is an easy way to find the place you are looking for.

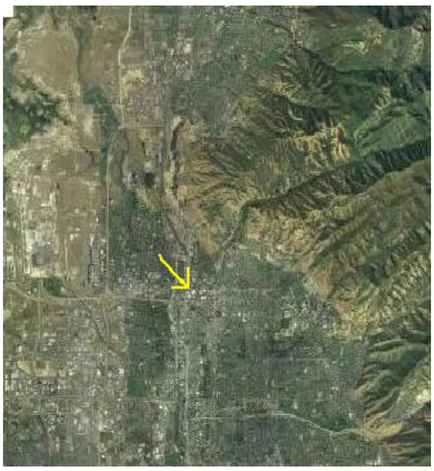
1. In the center of the webpage there will be an international coverage map which shows where TerraServer contains imagery for the world (highlighted green).

   NOTE: Most of the data will pertain to the USA and close neighbors.
2. For this walkthrough, click on the USA. The coverage map will zoom in.
3. Click on the map again on the state of Utah.

   NOTE: You are now zoomed all the way in on the coverage map. The next click will take you to the images on The National Map.

   HINT: If you are looking for another area there are a few tools that you may use to get there from here.
4. Click on the most likely place you believe Salt Lake City to be. Now you are looking at The National Map on TerraServer. Here you will be able to view topo maps, and black and white aerial photos. If available you may also view color DOQs.

   NOTE: Navigation here is tricky and difficult because there are no location labels on the map. But there is a location name and most recent photo indicator above the map window.
5. If the title indicates Salt Lake City, Utah, then you're close enough. If not then pan around until you get Salt Lake City. The image should look something like that below.



6. After you have found the image above zoom in on the white dot indicated by the arrow.

7. Continue zooming in until you recognize that it is the Delta Center. You can get very close.

8. Now click on the **Download** link in the top upper-right hand border of the map window.
9. Right click anywhere on the map/photo image and choose **Save Picture As....**
10. Change the filename to something more recognizable (example: deltacenter), and save it somewhere to your computer you can find easily.
11. You have the image file, now you need the world file. Click on the **World File** link in the top upper-right hand border of the map window, (it is in the same row as the Download link).
12. Once again save the file to an easily accessible location on your computer.

    HINT: It is best to save the world file to the same location as the image file. Also rename the world file to the same as that of the image file and change the extension to *.jpw. For example, if you saved the image file as deltacenter.jpg, then name the world file deltacenter.jpw. Upon importing the image file, some software simultaneously looks for a world file with the same filename prefix in order to automatically geo-reference the image.

You now have all the relevant files in order to import the image into WMS.

Using the TerraServer USA search panel

The search panel is located in the left-hand column on the TerraServer site.

"PICTURE"

It is a quick find search engine. All you really need to do is type in keywords pertaining to the street, city, and state address names for your area of interest. If you know the coordinates of the place you may also type those in to find the location.

Street, City, State address search:

1. For a pass-through type "main" in the Street field, "salt lake city" in the City field, and "utah" in the State field. Hit the GO button next to the State field. A list of matching address choices will pop up on an Advanced Find page.

    "PICTURE"

    NOTE: Imagery available for the matches is listed on the right side of the page.
2. Investigate your choices to see if any match where you are searching for. Find N Main St, Salt Lake City, UT 84103. Click on the Urban Areas image link. You will be taken directly to The National Map, zoomed in as close as possible, with a red pin symbol indicating the exact address match.

    NOTE: Your location of interest may not be listed. Try different keywords in the field boxes for varying results.

3. Zoom out two steps to get a wider view of the area.
4. You have found Temple Square in Salt Lake City.

Coverage Map search:

It is possible to click on the miniature coverage map in the search panel. By doing so you will be immediately taken to The National Map. From there you may navigate as you please.

Longitude, Latitude coordinate search:

If you happen to know the decimal degree coordinates of a place you can type them into the field boxes and hit the GO button. You will be taken directly to the specified coordinates on The National Map where you can navigate further.

NOTE: Wherever you are on The National Map a yellow dot will appear in the corresponding spot on the miniature coverage map of the US in the search panel. Similarly, the coordinates of the location will also appear in the Latitude - Longitude field boxes.

Using the Advanced Find search on TerraServer USA

The Advance Find search tool is in the toolbar at the top of the website.

"PICTURE"

There are four options in the Advanced Find tool:

1. Address
2. Place
3. Decimal Degrees
4. DMS (Degrees, Minutes, Seconds)

"PICTURE"

Address search:

Works simply by typing in the relevant name for Street, City and State fields. Also includes a Zip Code field for matching. Once entered, selecting the GO button will process the request and return a list of matching addresses. Image data accessible will be linked to each choice.

HINT: If your place does not come up vary the keywords in the search.

Place search:

Type in the common place name and a list will be produced with all matching references.

HINT: If your place does not come up vary the keywords in the search.

Decimal Degrees search:

If you have the latitude and longitudinal coordinates of the point in question, input them and you will be taken directly to the National Map when you hit the GO button.

NOTE: Remember to reference the degrees properly from the prime meridian (west-negative, east-positive), and the equator (north-positive, south-negative).

DMS search:

Similar to the previous method, but different in how the coordinates are input.

NOTE: Be sure to distinguish the correct direction latitudinal (north-south), and longitudinal (east-west).

7.2.c.4. Meteorologic Data

GSDA:Meteorologic Data

Maps, Tables, and Charts of Snow and Rainfall

Obtain NEXRAD Radar Data from NCDC [1]

- Help – Obtaining NEXRAD Radar Data from NCDC

http:/ / wikis. aquaveo. com/ xms/ FTPImages/ GSDA/ GSDA_NOAA_NESDIS. png [13] Obtain historical precipitation data from the NNDC Online site [2]

Fifteen minute and hourly historical data in ascii format at no cost.

- Help – Obtaining precipitation data from the NNDC

http:/ / wikis. aquaveo. com/ xms/ FTPImages/ GSDA/ GSDA_NOAA. png [13] Obtain digital rainfall grids from NOAA Atlas 2 [3]

Precipitation-Frequency maps for the Western US at a 15 ' resolution (~400m)

- Grid in Geographic NAD 83 coordinates
- Grid values in inches * 100,000

- Help – Obtaining a digital rainfall grid from NOAA Atlas 2

http:/ / wikis. aquaveo. com/ xms/ FTPImages/ GSDA/ GSDA_NOAA. png [13] Obtain precipitation data from NOAA [4]

A massive collection of climate-related data, tables and maps for gages, states and more.

- Help – Obtaining precipitation data from NOAA

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA_RCC.png [5] Obtain precipitation data from the RCC [6]

A variety of precipitation data from NOAA Atlas 2 precipitation/duration/frequency maps to snow data.

- Help – Obtaining precipitation data from the RCC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_UCC.png [7] Obtain precipitation data from the Utah Climate Center [8]

Gage data (i.e., precipitation, snowfall, temperature) for all 50 US states and around the world.

- Help – Obtaining precipitation data from the UCC

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png [9] Obtain precipitation data from NRCS PRISM [10]

Thirty-year average (normal) precipitation maps for each state as well as GIS precipitation data.

- Help – Obtaining precipitation data from NRCS PRISM

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png [9] Obtain precipitation data from NRCS NWCC [11]

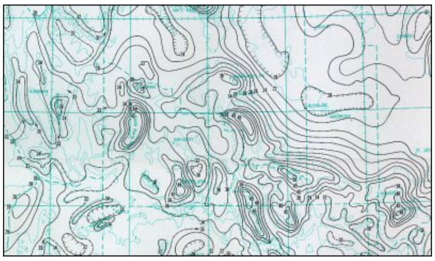
A wide variety of precipitation and snow data in graph, chart and table format.

- Help – Obtaining precipitation data from the NRCS http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_OneRain.png [12] Obtain precipitation data from OneRain [12]

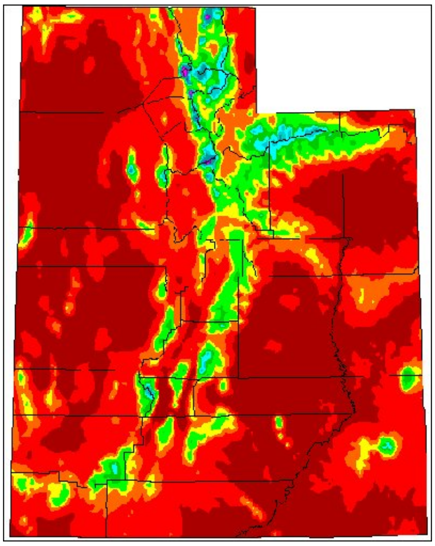
Radar and rain gage data combined to create accurate rainfall estimates. This data must be purchased.

Precipitation Overview

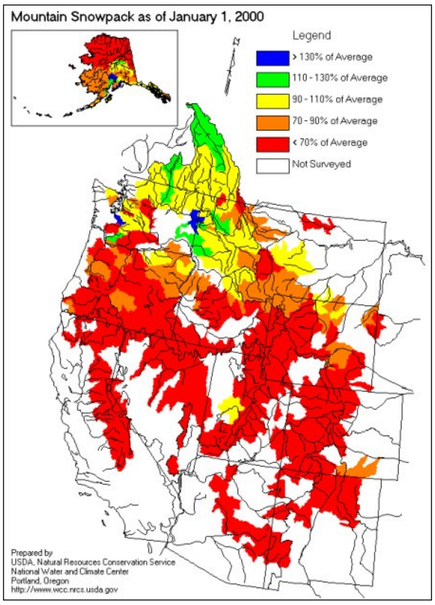
Climatological data such as precipitation, evapo-transpiration, wind and temperature are usually found in university libraries and are published by the U.S. Department of Commerce National Oceanic and Atmospheric Administration (NOAA). Precipitation data includes depth, intensity, duration, recurrence intervals, snowfall etc. Methods of calculating precipitation for a watershed from gage data include the Thiessen polygon method, the isohyetal method, and a simple arithmetic mean. Other forms of precipitation such as snow are also important. Snow data can be collected by various means such as SNOTEL and snow course.



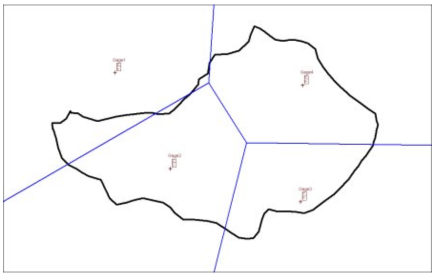
Precipitation map from NOAA Atlas 2 showing a portion of Arizona for the 100-year 6-hour storm event.



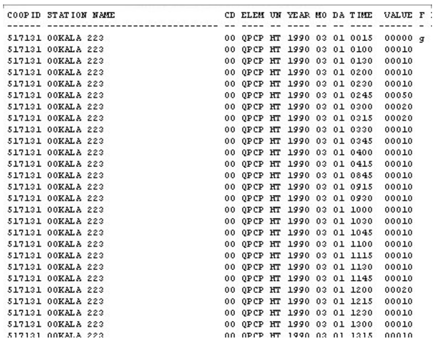
Normal annual precipitation for Utah (legend not shown) obtained from PRISM.



Snowpack data from the NRCS showing the western US.



Thiessen polygons can be created for a watershed based on gages locations.



COOPID	STATION NAME	CD	ELEM	UN	YEAR	MO	DA	TIME	VALUE	F
517131	00KALA 223	00	QPCP	HT	1990	03	01	0015	00000	g
517131	00KALA 223	00	QPCP	HT	1990	03	01	0100	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0130	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0200	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0230	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0245	00050	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0300	00020	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0315	00020	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0330	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0345	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0400	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0415	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0845	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0915	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	0930	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1000	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1030	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1045	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1100	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1115	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1130	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1145	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1200	00020	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1215	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1230	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1300	00010	
517131	00KALA 223	00	QPCP	HT	1990	03	01	1315	00010	

Historical data can be obtained on the internet and at university libraries.

Precipitation Tips

- How do I obtain precipitation data from NNDC Climate Data Online
- How do I obtain precipitation data from NOAA
- How do I obtain precipitation data from RCC
- How do I obtain precipitation data from the UCC
- How do I obtain precipitation data from NRCS PRISM
- How do I obtain precipitation data from NRCS NWCC
- How do I decompress data files

References

[1] http://www.ncdc.noaa.gov/oa/radar/radardata.html
[2] http://hurricane.ncdc.noaa.gov/CDO/cdo
[3] http://www.nws.noaa.gov/oh/hdsc/noaaatlas2.htm
[4] http://www.noaa.gov/climate.html
[5] http://www.wrcc.dri.edu/
[6] http://www.wrcc.dri.edu/rcc.html
[7] http://climate.usurf.usu.edu/
[8] http://climate.usurf.usu.edu/products/data.php
[9] http://www.nrcs.usda.gov/
[10] http://www.wcc.nrcs.usda.gov/climate/prism.html
[11] http://www.wcc.nrcs.usda.gov/
[12] http://www.onerain.com/

GSDA:Obtainig NEXRAD Radar Data from NCDC

Obtaining NEXRAD Radar Data from the National Climatic Data Center (NCDC) requires two steps. First, you must order the data. Second, you need to convert the data to an ASCII grid format so WMS can recognize it. It is available from September 1996 until a few days from the present (availability does depend on the radar site).

Ordering Data from NCDC

There are two methods to order the data from NCDC. One method is using the 'NEXRAD Inventory Search Tool' [1]. The other method is to order the data from the 'NCDC HDSS Access System (HAS) [2]. Method 1 is the simplest method if you need a single day of data for a single station.

Method 1: Using the NEXRAD Inventory Search (Recommended Method)

This method provides a map to help you determine the radar site closest to your watershed and will not require you to know the 4 digit acronym of the radar site (For example, Salt Lake City is KMTX. This is provided when a radar site is selected). It allows you to see the data that is available before making the order. It only allows you to order data one day at a time. It also allows you to order a specific product ("LVL3 - STORM TOTAL PRECIP (230 KM)" in our case) and select the files (timesteps) you wish to order.

1. Select the radar site closest to your watershed from 'National Doppler Radar Sites' [1].
2. Choose the date of the storm events that interests you.
    1. Note the reference timezone, and that you may need to convert your date.
3. Select "LVL3 - STORM TOTAL PRECIP (230 KM)" for the product.

4. Click Create Graph button.
   1. This graph will show which mode that the radar was operating under when collecting the data. The red or blue lines indicate times that data was collected. Typically this is around every four minutes, but occasionally the radar may be done for several hours or longer.
5. Click the Order Data hyperlink.
6. Enter your Email Address.
7. Select the files you wish to download.
   1. The file name format is explained on the NCDC Radar Frequently Asked Questions (FAQ) [3]. Most importantly, the last block of numbers gives the year, month, day, hour, and minute that the data was collected.
8. Click Order Data button.
9. The final page will confirm that your order has been submitted and provide you a link to check on the status of your order. It generally takes less than 5 minutes to process your data and make it available on the ftp site. You can wait to receive the e-mail which will provide you with the ftp link to download the data, or you can use NCDC Java NEXRAD Viewer and Data Exporter [4] to download your data. The details of how to use the Java NEXRAD Viewer and Data Exporter will be given in the convert data step

Method 2: NCDC HDSS Access System (HAS) (Advanced Method)

This method requires you to know the 4 digit acronym of the radar site that is closest to your watershed; however, you can download a Google Earth kmz file from the NCDC for this purpose: [www.ncdc.noaa.gov/oa/radar/nexrad.kmz Nexrad.kmz]. It only allows you to order data that may span days, months, or years. It will send all the products available for the specified time period in tar file format.

1. Click on NEXRAD Level III Radar image hyperlink at the NCDC HDSS Access System (HAS) [2] site.
2. Select the Radar Station(s) of interest.
3. Select Start and End Date.
4. Enter your E-mail Address.
5. Click the Continue with Selections button.
6. Click the Retrieve Selected Files button.
7. The final page will confirm that your order has been submitted and provide you a link to check on the status of your order. It generally takes less than 5 minutes to process your data and make it available on the ftp site. You can wait to receive the e-mail which will provide you with the ftp link to download the data, or you can use NCDC Java NEXRAD Viewer and Data Exporter [4] to download your data. The details of how to use the Java NEXRAD Viewer and Data Exporter will be given in the convert data step

Convert the data to an ASCII grid format

The files obtained from the NCDC are given in a binary format that is foreign to WMS. So we will now convert them to a format that WMS can read.

1. From the 'Java NEXRAD Tools' [4] webpage, click on the *Install Now hyperlink*.
2. Click on the Launch the STABLE Version 1.9.7 Java NEXRAD Viewer (includes Data Exporter) Hyperlink located under the Java Web Start heading. This will install the Java program on your machine if it is the first time you have run the program. Subsequent visits will launch the program rather than installing it.
3. If you wish to use the Java NEXRAD Viewer to download your data from the NCDC, enter your HAS number in the Data Selector dialog and click submit (The HAS number was given on the confirmation page and e-mail). If you have already downloaded your data, change the tab on the Data Selector dialog from NCDC to Local. Then click Browse, and select the folder that contains your data.
4. You can now view your data by selecting a file, and clicking the Load button.

5. In the Java NEXRAD Viewer Main window, click on the Data Menu, and select NEXRAD Data Exporter or hit Ctrl-E.
6. The Java NEXRAD Data Exporter should have brought up your data in its list box. If it did not, follow the same routine you did earlier for the Data Selector dialog.
7. Click on the Format Menu, and Select ESRI ASCII Grid file format.
8. It is recommended that you change the grid size of the exported ASCII Grid.
    1. Click on the Options Menu and select Raster Options.
    2. You can Change either the Square Grid Size or Grid Cell Size (deg) but the other will need to be set to Auto. Choosing a large value will reduce the time required by WMS to process the data. Choosing the a small value will give you more resolution. It is recommended that you increase the size of the grid cells.
        1. A Grid Cell Size (deg) of 0.0100 will yield roughly 20 points in a basin of 7 square miles (depending on shape) while a Grid Cell Size (deg) of 0.0250 will yield roughly 400 points in a basin of of 1,000 square miles (again, depending on shape).
    3. Click the OK button.
9. Select the files you wish to export.

If you used Method 2 to collect your data, you will see all the data products that are available instead of only the Total Storm Precipitation. Be sure to only select the files that have NTP in the product designation. More information on filename format in the NCDC Radar Frequently Asked Questions (FAQ) [3].

1. Click the Browse button and select the folder you wish to place the exported ASCII grids.
2. Click the Export button.
3. When the program finishes, the data is ready to be brought into WMS.

Metadata

About the Data

The data you have now downloaded and converted gives the cumulative depth of the precipitation in inches since the beginning of the storm (the data and time of the beginning of the storm is shown when the data is loaded in the NCDC Java NEXRAD Viewer and Data Exporter [4]. The data is given in geographic coordinates. A projection file (prj) is included for viewing within ESRI ArcGIS. Projection files are not read by WMS. Because the data is cumulative, you do not need to use every timestep provided by the NCDC. It is recommended that you choose a regular interval (for example, an hour), and select the files that are the closest to these time intervals.

Using Radar Data in WMS

Refer to the WMS documentation

References

[1] http://www.ncdc.noaa.gov/nexradinv/
[2] http://has.ncdc.noaa.gov/pls/plhas/has.dsselect
[3] http://www.ncdc.noaa.gov/nexradinv/faq.html#FORMAT
[4] http://www.ncdc.noaa.gov/oa/radar/jnx/index.html

GSDA:Obtaining Precipitation Data from NOAA

NOTE: NOAA has collected such a massive amount of climatological data that only a few resources are pointed out in this discussion. Feel free to browse the NOAA web site and/or contact NOAA staff for more information about any given data type.

Example 1

Go to the NOAA National Climatic Data Center at http://lwf.ncdc.noaa.gov/oa/ncdc.html.If this link does not work, follow the bulletted steps below:

- Go to the NOAA homage at www.noaa.gov.
- Click on the *Climate* link near the bottom.
- Click on the *National Climatic Data Center* link.

Much of the data here is normally bound in several volumes of books and found at most universities. Data such as precipitation, evaporation, wind, and temperature are recorded monthly and annually. This information is free online to education but often comes with a cost for other purposes.

1. Choose the *Most Popular Products* link.
2. Choose the *Publications* link.
3. Choose the *Climatological Data* link.

If you require this data for commercial purposes:

1. Choose a state to and click the *Select A State* button.
2. Scroll through the list of available dates and add the desired item(s) to your "shopping cart".

If you require this data for educational purposes:

1. Click the *publications* link near the bottom of the page.
2. Choose the *Climatological Data* link.
3. Choose the *FREE access by certain agencies and individuals* link.
4. Choose a state from the list.
5. Click on the desired year.
6. Click on the file within that year's folder and proceed to download it. The files are typically Adobe Acrobat (.pdf) files and can only be viewed with the Acrobat Reader (which is available for free download at www.adobe.com/products/acrobat/readstep.html [1].
7. The data is typically several pages long, but see the image below for a sample of what it may look like:

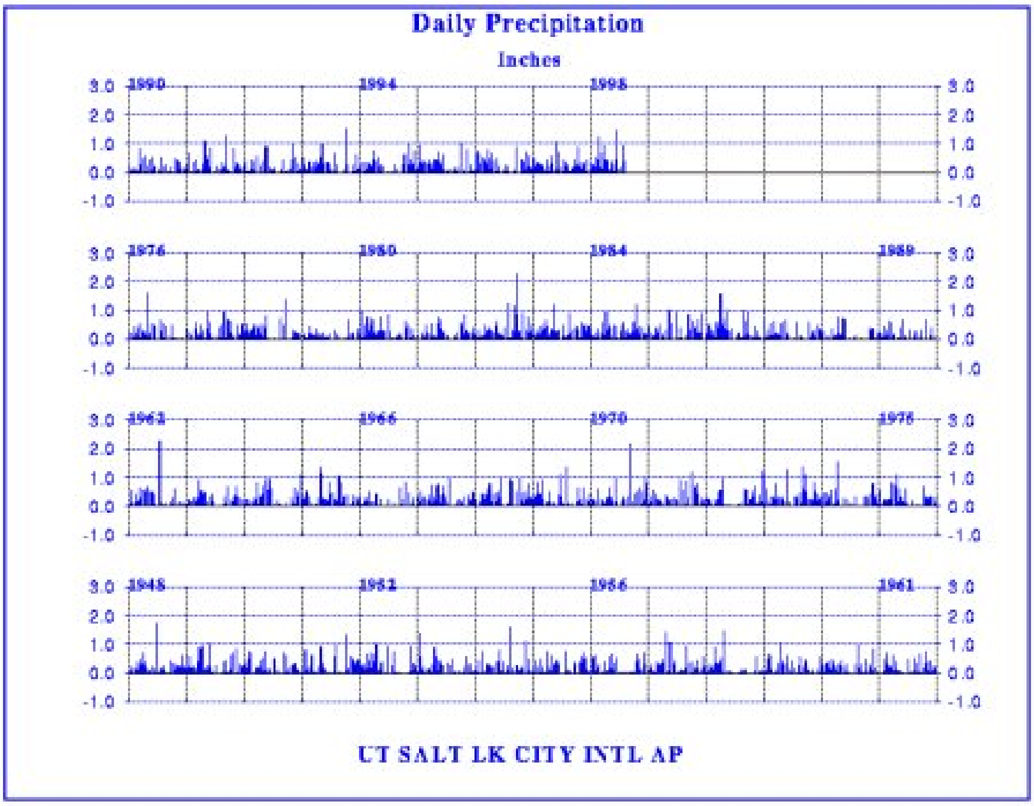
UTAH
JANUARY 2000

MONTHLY STATION AND DIVISION SUMMARY

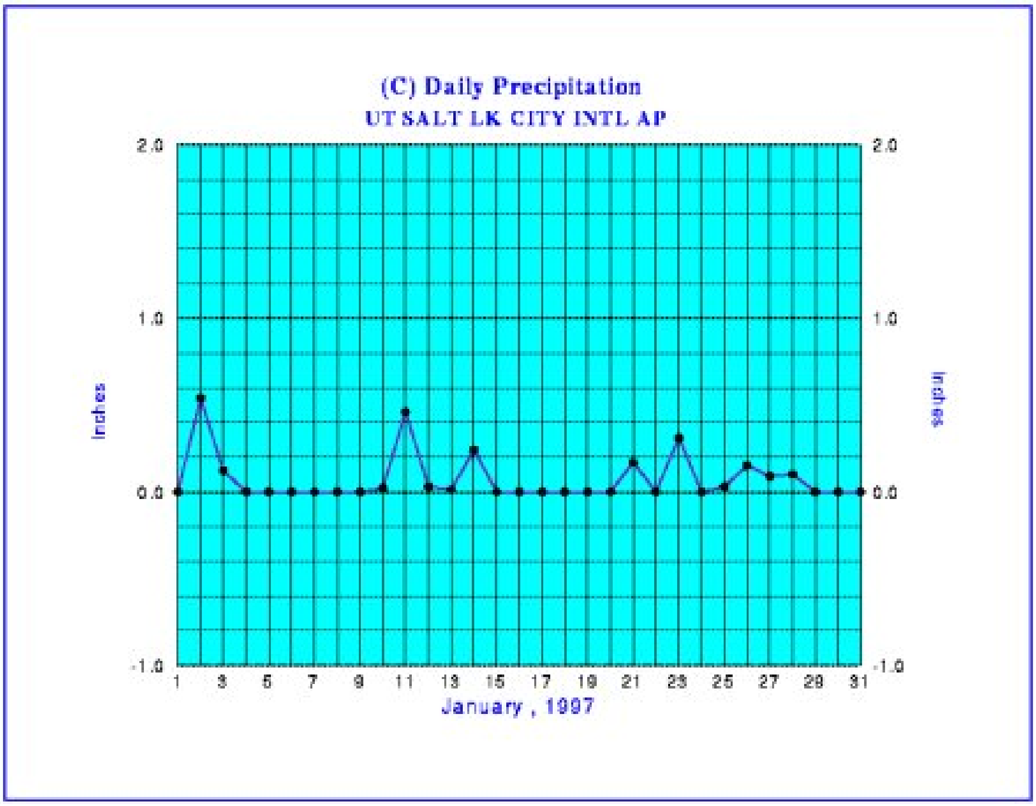
STATION		TEMPERATURE (°F)										NO. OF DAYS			
												MAX		MIN	
		AVERAGE MAXIMUM	AVERAGE MINIMUM	AVERAGE	DEPARTURE FROM NORMAL	HIGHEST	DATE	LOWEST	DATE	HEATING DEGREE DAYS	COOLING DEGREE DAYS	90 OR ABOVE	32 OR BELOW	32 OR BELOW	0 OR BELOW
UTAH															
WESTERN 01															
BLACK ROCK		47.9	23.4M	35.7M	9.2	63	15	-3	6	908	0	0	0	23	3
CALLAO		48.8	21.0	34.9	9.0	64	14	4	6	926	0	0	0	25	0
DELTA		47.0	22.9	35.0	10.1	59	18	2	6	925	0	0	0	24	0
DESERET		46.4	25.3	35.9	11.6	62	18	5	6	894	0	0	1	23	0
DUGWAY	B	46.7	20.8	33.8	8.2	64	11	4	6	960	0	0	0	27	0
ENTERPRISE BERYL JCT		47.1	16.6	31.9	6.5	62	21+	-12	3	1021	0	0	3	30	6
ESKDALE		49.5	23.0	36.3		64	11	-1	6	885	0	0	0	22	1
FISH SPRINGS REFUGE		47.5	23.9	35.7	9.1	65	17	8	6	902	0	0	0	26	0
GROUSE CREEK		37.8	16.5M	27.2M	5.1	48	21+	0	6	1163	0	0	7	30	1
IBAPAH		49.5	15.7	32.6	7.3	62	18	0	6	997	0	0	0	31	1
KNOLLS 10 NE	B	42.2	22.4	32.3		61	17	5	6	1007	0	0	0	26	0
MILFORD		M	M	M											
MODENA		M	M	M											
PARTOUN		48.1	21.0	34.6	8.6	63	16	0	6	937	0	0	0	28	1
ROSETTE		38.6	21.4	30.0		55	17	9	6	1077	0	0	6	30	0
UTAH TEST RANGE		43.3	23.2	33.3		61	16+	8	7+	978	0	0	0	27	0
WAH WAH RANCH		M	M	M											
WENDOVER AMOS	B	40.8	25.1M	33.0M	6.2	63	11	12	8+	985	0	0	3	23	0
--DIVISIONAL DATA------->				33.5	7.5										
DIXIE 02															
LA VERKIN		57.7	32.4	45.1	6.2	72	20	18	8	611	0	0	0	15	0
LYTLE RANCH		60.5	26.3M	43.4M		73	18	13	5	667	0	0	0	22	0
ST GEORGE		M	M	M											
VEYO POWERHOUSE		51.0	30.6	40.8	5.6	63	20	17	4	745	0	0	0	16	0
ZION NATIONAL PARK		57.5	34.7	46.1	5.9	71	20	11	3	579	0	0	0	12	0
--DIVISIONAL DATA------->				43.9	5.1										
NORTH CENTRAL 03															
ALPINE		43.3	28.5	35.9	9.3	57	16+	12	6	896	0	0	4	19	0
BRIGHAM CITY WASTE PLT		41.2	25.0	33.1		57	16	8	7	980	0	0	4	25	0
COTTONWOOD WEIR	//	44.2	28.7	36.5	6.7	60	16+	15	3	880	0	0	2	21	0
CUTLER DAM UTAH P&L CO		39.9	26.6	33.3		49	21	13	6	978	0	0	4	23	0
DRAPER POINT OF THE MT		46.5	30.8M	38.7M		61	16+	14	6	805	0	0	1	16	0
FAIRFIELD		44.4	21.9	33.2	9.7	58	11	1	6	979	0	0	1	26	0
FARMINGTON COURTHOUSE		42.4	26.3	34.4	5.8	58	17	14	1	941	0	0	3	23	0
GARFIELD		43.4	31.3M	37.4M	8.1	62	16	22	6	845	0	0	1	17	0
GRANTSVILLE 2 W		43.8	25.2M	34.5M		61	17	14	1	935	0	0	1	26	0
JOHNSON PASS		43.9	22.7	33.3		57	16+	6	3	976	0	0	2	26	0
LITTLE SAHARA DUNES		44.2	20.4M	32.4M		56	18	2	6	1011	0	0	2	22	0

Example 2

1. Return to the NOAA web site at http://lwf.ncdc.noaa.gov/oa/ncdc.html.
2. Choose the *Most Popular Products* link.
3. Choose the *Free* link.
4. Click the *Climate Visualization* link.
5. Select the *Time Series* link under the "National Weather Service Summary of the Day" heading.
6. You have several choices for data acquisition here. For this example click on the *Display the Period of Record for a Parameter at one Station* link.
7. Select *Utah* as the state and click the *Continue to Next Page* button.
8. Select a weather station from the list. Scroll down near the bottom of the list and select station *24127, UT SALT LK CITY INTL AP*.
9. Select *(C) Daily Precipitation* from the *Select the Parameter* list.
10. You may choose "GIF" or "Postscript" as the *Output Format*. For this example choose GIF.
11. Click on the *Submit Graph Values* button.
12. In this case, nearly 60 years of data are displayed in graphical form (see figure below).



1. Return to http://lwf.ncdc.noaa.gov/oa/climate/onlineprod/tfsod/climvis/main.html.
2. Choose the *Display One Parameter for One Station for a Specified Time Frame* link.
3. Select *Utah* as the state and click the *Continue to Next Page button*.
4. Select a weather station from the list. Scroll down near the bottom of the list and select station *24127, UT SALT LK CITY INTL AP*.
5. Select *(C) Daily Precipitation* from the *Select the Parameter* list.
6. Enter "1997" in the *Enter the Year* field.
7. In the *Select Range of Months* field, specify January through January (just to see the month of January).
8. Choose *GIF* as the *Output Format*.
9. Click on the *Submit Graph Values* button.
10. The graph should look similar to the one below.



References

[1] http://www.adobe.com/products/acrobat/readstep.html

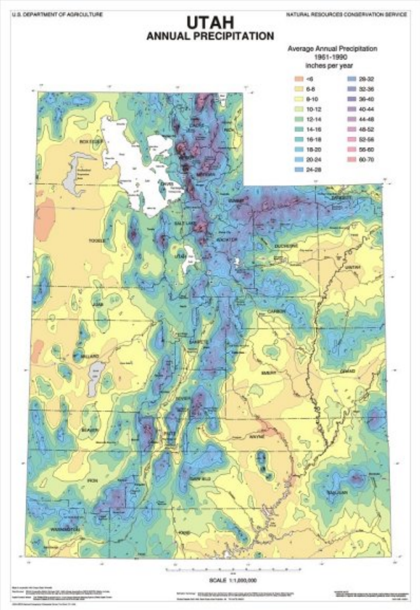
GSDA:Obtaining Precipitation Data from PRISM

PRISM data (Parameter-elevation Regressions on Independent Slopes Model) provides maps of each state with its average annual precipitation. You may purchase precipitation maps in addition to downloading them.

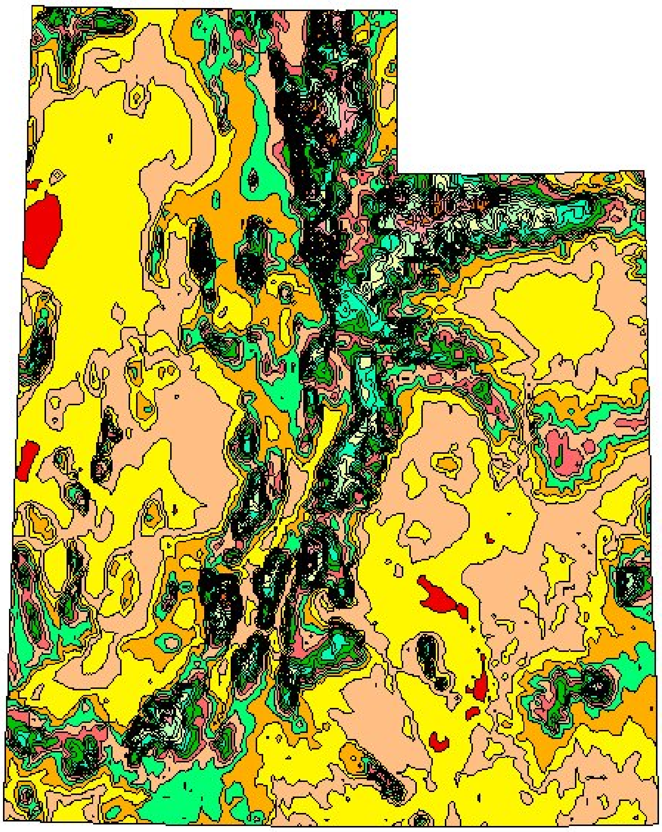
Go to the USDA PRISM web site at www.wcc.nrcs.usda.gov/water/climate/prism/prism.html [1]. If you have trouble with this link, follow the bulletted steps below:

- Go to the NRCS homage at http://www.nrcs.usda.gov,
- Click on the *Technical Resources* button at the page top,
- Scroll down and click on the *Water Resources* link,
- Choose the *Water and Climate Data* link,
- Click the *Climate* link,
- Choose the *Spatial Climate Products (PRISM)* link.

1. Choose the *Cartographic-Quality Maps* link. Note that there are also a link for *Gridded Precipitation Maps*. The first link gives average annual precipitation for the different US states.
2. Choose *Maps* near the top of the screen.
3. Choose a state by its two-letter abbreviation.
4. You will see a small thumbnail image of the state's precipitation map. If you are going to print the map to a large format plotter, choose the Map ps link (post script files are very large (e.g., 32 MB for Texas) but can also be read by typical graphics programs such as Paintshop Pro. Otherwise, a color Adobe Acrobat (.pdf) map of the state can be downloaded or viewed by clicking the Map pdf link.



1. Backtrack to the main PRISM website at www.wcc.nrcs.usda.gov/water/climate/prism/prism.html [1].
2. Again choose the *Cartographic-Quality Maps* link.
3. Click on *Data* near the top of the screen. Precipitation data for different areas (i.e., states, regions, country) are available.
4. For example, click on the *State* link.
5. Scroll down until you find the state you need data for.
6. Each state offers its data in ARC/INFO, ArcView, and ASCII formats - click the appropriate link. The ArcView precipitation data for Utah is shown below.



References

[1] http://www.wcc.nrcs.usda.gov/water/climate/prism/prism.html

GSDA:Obtaining Precipitation Data from the NCDC / NNDC Climate Data Online

15-minute and hourly precipitation data can be downloaded from the National Climatic Data Center's NNDC Climate Data Online website. The site has data for 6,100 U.S. stations, and some non-U.S. stations.

To download 15-minute or hourly precipitation data

1. Go to the NNDC Climate Data Online [1] website.
2. Select "Precipitation Data, 15 minute" (or "Precipitation Data, Hourly") under the *Select a dataset* section.
3. Click the Continue button.

   Wait a few moments for the next page to load.
4. Select the Country option and change its field to "United States".
5. Click Continue.
6. Select your desired state under the select a *State/Province* section.
7. Choose the option for *Selected Station in the state*.
8. Click Continue.
9. Select the station(s) of choice, noting the period of record for each station.

   You can multi-select stations by holding the Ctrl key when clicking.
10. Click Continue.
11. Enter the date criteria and desired output format.

    The space-delimited format works nicely if you want to view the data in your browser.
12. Click Continue.
13. Enter a valid email address.
14. Click the Submit Request button.
15. Make a note of the URL provided in the ensuing web page.

    When the data request has been processed, the data can be accessed at the URL provided. You will be notified when the data is ready at the email address you entered.
16. Click on the URL.
17. Click on the first Access URL, which links to the data file.

We suggest you read the dataset documentation, which is the last access link.

The 15 minute documentation states that all precipitation values are in hundredths of inches (value X 0.01 = inches). You can check the accuracy of the precipitation observation by viewing the entry in the column labeled "UN." If it is HT, then the values are in stored to the tenths place. If it is HI, then the values are stored to the hundredths place.

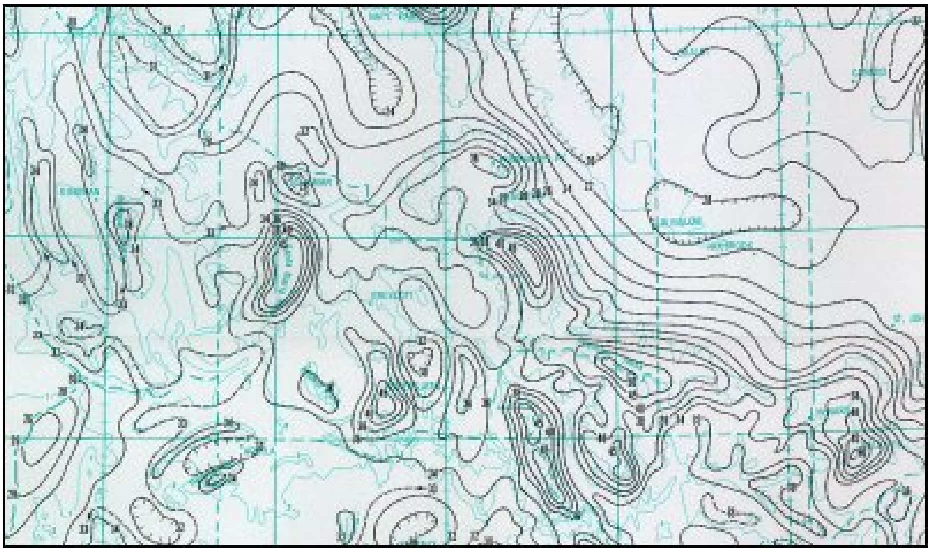
References

[1] http://cdo.ncdc.noaa.gov/plclimprod/plsql/poemain.poe

GSDA:Obtaining Precipitation Data from the RCC

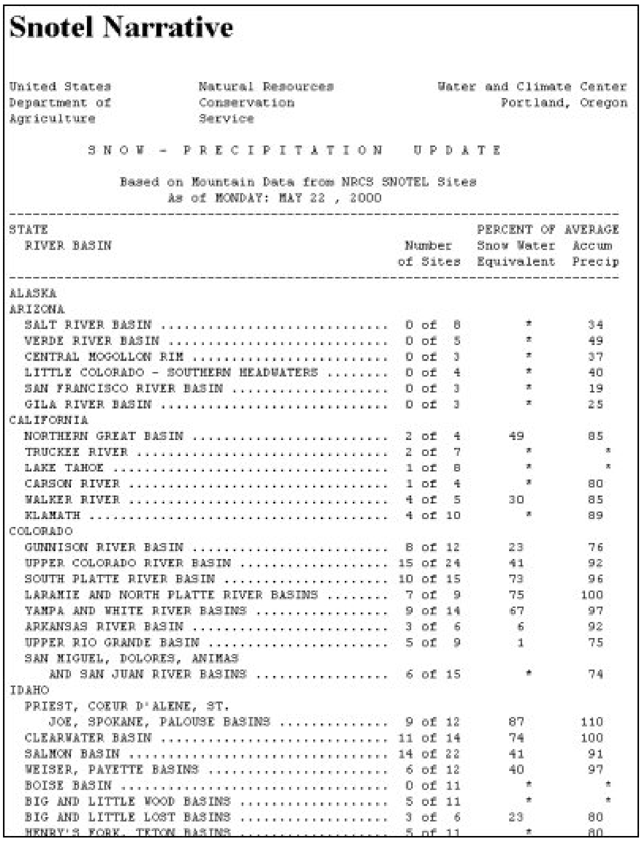
Example 1

1. Go to the Regional Climate Center's web site at www.wrcc.dri.edu/rcc.html [6].
2. Once you arrive at the Regional Climate Center (RCC) web site, select the desired region from the map. For this example, choose the western region in yellow.
3. Under the "Precipitation Maps" section (on the right), you have two choices. Feel free to explore both of them. For example, click on the Precipitation Frequency Maps for the Western U.S. (Return Periods. NOAA Atlas 2) link.
4. Choose a state, return period, and storm duration. For example, click on the link for the "100-year 6-hour" storm for Arizona.
5. The map will display similar to what is shown below (only a portion of the map is shown).



Example 2

1. Return to the Regional Climate Center's web site at www.wrcc.dri.edu/rcc.html [6].
2. Once you arrive at the Regional Climate Center (RCC) web site, select the desired region from the map. For example, choose the western region in yellow.
3. Choose the SNOTEL Data (Oct. 93 till Today is available.) link at the left.
4. There are several things you can do on this page, but for this example click the Latest West-wide Snotel basin and station conditions from NRCS link.
5. A "Snotel Narrative" will be listed similar to the image below:



Snotel Narrative

United States Department of Agriculture — Natural Resources Conservation Service — Water and Climate Center, Portland, Oregon

S N O W - P R E C I P I T A T I O N U P D A T E

Based on Mountain Data from NRCS SNOTEL Sites
As of MONDAY: MAY 22 , 2000

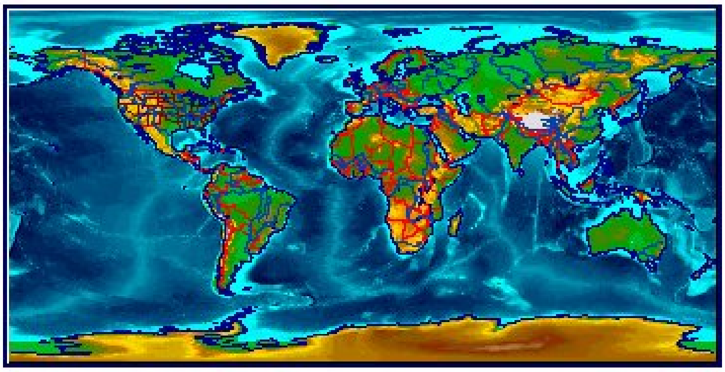
STATE / RIVER BASIN	Number of Sites	PERCENT OF AVERAGE Snow Water Equivalent	PERCENT OF AVERAGE Accum Precip
ALASKA			
ARIZONA			
SALT RIVER BASIN	0 of 8	*	34
VERDE RIVER BASIN	0 of 5	*	49
CENTRAL MOGOLLON RIM	0 of 3	*	37
LITTLE COLORADO - SOUTHERN HEADWATERS	0 of 4	*	40
SAN FRANCISCO RIVER BASIN	0 of 3	*	19
GILA RIVER BASIN	0 of 3	*	25
CALIFORNIA			
NORTHERN GREAT BASIN	2 of 4	49	85
TRUCKEE RIVER	2 of 7	*	*
LAKE TAHOE	1 of 8	*	*
CARSON RIVER	1 of 4	*	80
WALKER RIVER	4 of 5	30	85
KLAMATH	4 of 10	*	89
COLORADO			
GUNNISON RIVER BASIN	8 of 12	23	76
UPPER COLORADO RIVER BASIN	15 of 24	41	92
SOUTH PLATTE RIVER BASIN	10 of 15	73	96
LARAMIE AND NORTH PLATTE RIVER BASINS	7 of 9	75	100
YAMPA AND WHITE RIVER BASINS	9 of 14	67	97
ARKANSAS RIVER BASIN	3 of 6	6	92
UPPER RIO GRANDE BASIN	5 of 9	1	75
SAN MIGUEL, DOLORES, ANIMAS AND SAN JUAN RIVER BASINS	6 of 15	*	74
IDAHO			
PRIEST, COEUR D'ALENE, ST. JOE, SPOKANE, PALOUSE BASINS	9 of 12	87	110
CLEARWATER BASIN	11 of 14	74	100
SALMON BASIN	14 of 22	41	91
WEISER, PAYETTE BASINS	6 of 12	40	97
BOISE BASIN	0 of 11	*	*
BIG AND LITTLE WOOD BASINS	5 of 11	*	*
BIG AND LITTLE LOST BASINS	3 of 6	23	80
HENRY'S FORK, TETON BASINS	5 of 11	*	80

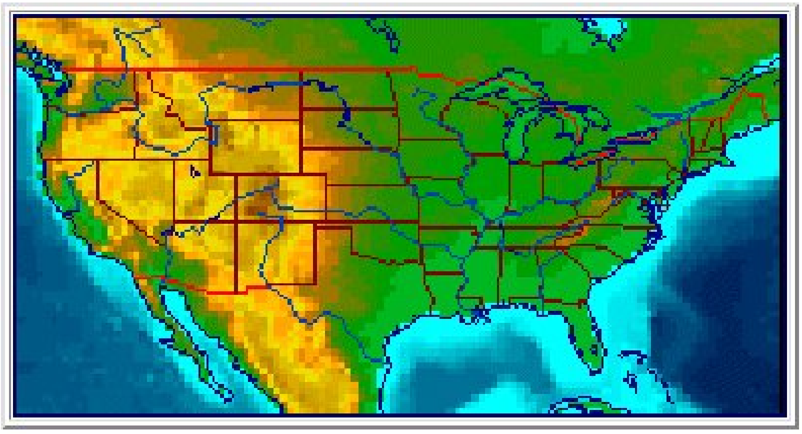
GSDA:Obtaining Precipitation Data from the UCC

Visit the Utah Climate Center (UCC) Weather Data Server at http://climate.usu.edu/weather/dataserv.htm.If this link does not work, follow the bulletted steps below:

- Go to the Utah State University (USU) Climate Center homepage at www.usu.edu [1].
- Click on the *Research* link near the bottom.
- Scroll down and click on the *Utah Climate Center* link.
- Under the *Weather Data* heading, click the *Data Server* link.

1. At this point you will see two maps: a world map above and a map of the United States below it (see figures below).
2. The map of the world will allow you to interactively click on any point and zoom in. When you are zoomed into the region of interest, click the *List Stations* button to list the weather stations *in that zoom box area only*.
3. The map of the U.S. will automatically list all stations in a particular state as soon as you click on the state.





1. For example, on the US map, click on the state of *Utah*.
2. Select a station from the alphabetical list. For example, select the *OREM TREATMENT PLANT* station.
3. Enter in the dates to view data for. In this case, view the entire year of 1998 by selecting the *Beginning Year* and the *Ending Year* as 1998.
4. Click the *Start* button.
5. You will see the following message: "Your data request is ready... 00040282.DAT"
6. Do a single right-click with the mouse and proceed to download the data file by choosing "Save Link As..." or "Save Target As...".
7. The downloaded .dat file may be imported into a spreadsheet program like Microsoft Excel. You may want to try a Delimited setting with Tab and Comma delimeters (see Microsoft Excel help for more information). With a little formatting you can have a nice looking spreadsheet. In this case, we have precipitation, snowfall, and temperature data for this station for the entire year of 1998 (the figure below shows a portion of this data).

Station #	Year	Month	Day	Total Precipitation (in.)	Total Snowfall (in.)	Snow depth on ground (in.)	Daily Max. Temperature (°F)	Daily Min. Temperature (°F)	Temperature at observation (°F)
42653003	1990	1	1	0	0	0	42	29	40
42653003	1990	1	2	0	0	0	49	39	47
42653003	1990	1	3	0.1	0	0	47	39	45
42653003	1990	1	4	0	0	0	47	40	45
42653003	1990	1	5	0.11	2	0	46	32	32
42653003	1990	1	6	0	0	0	33	24	32
42653003	1990	1	7	0	0	0	36	27	35
42653003	1990	1	8	0	0	0	41	30	39
42653003	1990	1	9	0	0	0	39	29	38
42653003	1990	1	10	0.11	0	0	42	38	42
42653003	1990	1	11	0.01	0	0	47	40	40
42653003	1990	1	12	0.06	1	0	46	36	41
42653003	1990	1	13	0.65	0	0	46	36	41
42653003	1990	1	14	0	0	0	42	32	40
42653003	1990	1	15	0.27	2	0	42	38	42
42653003	1990	1	16	0.07	0	0	43	36	43
42653003	1990	1	17	0	0	0	49	41	45
42653003	1990	1	18	0	0	0	47	38	44
42653003	1990	1	19	0.28	0	0	44	40	40
42653003	1990	1	20	0.06	0	0	41	35	39
42653003	1990	1	21	0.05	0	0	41	35	36

References

[1] http://www.usu.edu

GSDA:Obtaining Precipitation or Snow Data from USDA / NRCS

Go to the USDA/NRCS National Water and Climate Center web page at www.wcc.nrcs.usda.gov [1]. If you have trouble getting there, follow the bulletted steps below:

- Go to the USDA home page at www.usda.gov [2].
- Click on the *Agencies, Services & Programs* button near the top.
- Click on the *Natural Resources Conservation Service (NRCS)* link, located about mid-page.
- Choose the *Technical Resources* button at page top.
- Click on the *Water Resources* link near the bottom.
- Click the *Water and Climate Data* link.

Precipitation Data

1. Once at the *Water and Climate Data* web page, click on the link for *Water Supply Forecasts*.
2. Several valuable links are located here. For this example, we will explore just a few of them. First, click the *State Basin Outlook Reports* link.
3. Select a state. For example, choose *Utah* and pick any month (e.g., January).
4. Click the *Retrieve Selection* button.
5. Scroll down the list of available basins and choose one of them.
6. Choose the *Narrative and Graphs* link. Again, several options for snow, precipitation, and reservoir storage are here. Explore around a bit.
7. Return to the *Water and Climate Data* web page at www.wcc.nrcs.usda.gov/wcc.html [2].
8. Click on the *Water Supply Forecasts* button.
9. Choose the *Precipitation Reports* link.

10. Choose a state from the list. For example, Utah.
11. Click the *Submit* button. Two types of data are available: BAPR and BSPR. BAPR data displays the percent of average and percent of last year for monthly and year-to-date precipitation. BSPR displays the current, last year and average values for monthly and year-to-date precipitation as well a basin summary. All the data is in table/text format.

Snow Data

Once at the *Water and Climate Data* web page, click on the link for snow. Any of the snow reports (e.g., SNOTEL) list contain valuable information.

1. For this example, choose the *Daily SNOTEL Snow/Precipitation Update Reports* link.
2. Choose the *State Reports* link.
3. Select a state (for example, Utah) from the list and click the *Retrieve Selection* button.
4. Choose the *Select this line to receive the Precipitation Update for Utah from our FTP* site link. The SNOTEL data report will appear similar to the image below (depending upon the data).

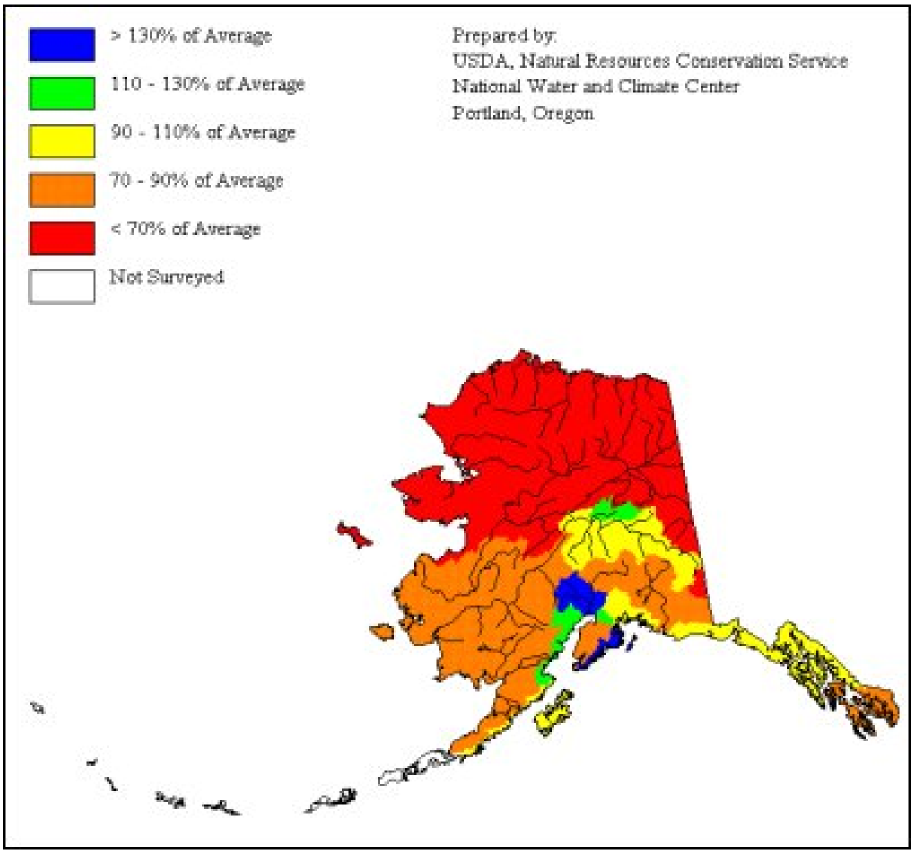
United States Department of Agriculture — Natural Resources Conservation Service — Water and Climate Center, Portland, Oregon

S N O W - P R E C I P I T A T I O N U P D A T E

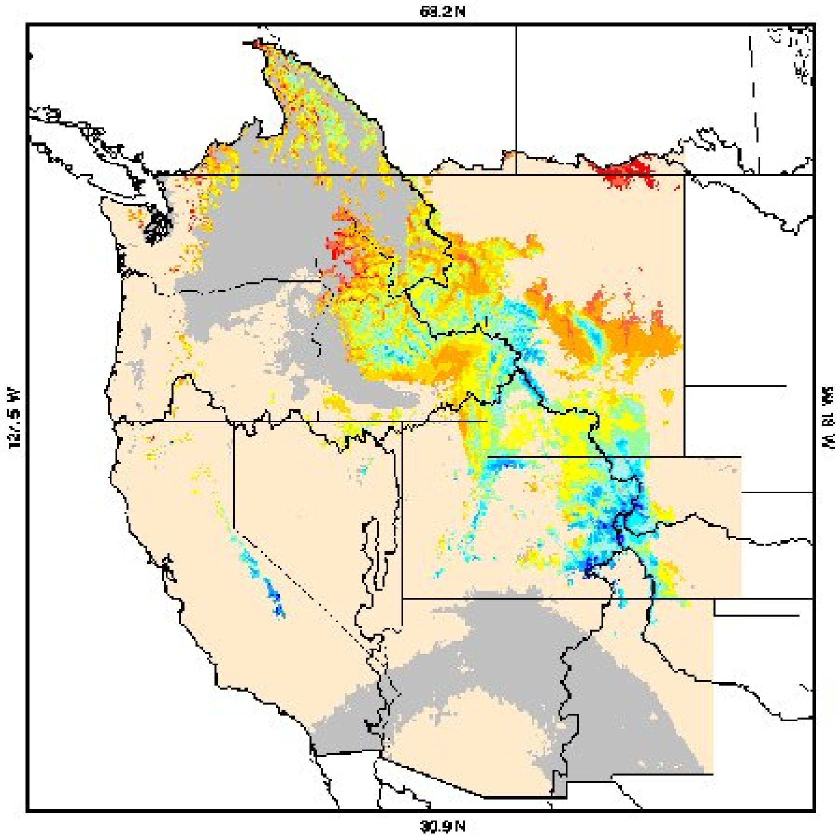
Based on Mountain Data from NRCS SNOTEL Sites
As of MONDAY: MAY 22 , 2000

BASIN / Data Site Name	ELEV. (Ft)	SNOW WATER EQUIVALENT Current	Average	% Avg	TOTAL PRECIPITATION Current	Average	% Avg
UTAH							
BEAR RIVER							
TRIAL LAKE	9960	9.9	17.3	57	24.4	30.6	80
HAYDEN FORK	9400	.0	.4	0*	20.6	25.2	82
LILY LAKE	9050	.0	3.8	0	16.6	20.1	83
MONTE CRISTO	8960	.7	16.9	4	23.5	32.6	72
TONY GROVE LAKE	8400	9.5	18.3	52	34.1	43.6	78
FRANKLIN BASIN	8040	2.0	13.6	15	30.7	39.7	77
BUG LAKE	7950	.0	6.9	0	16.9	24.2	70
LITTLE BEAR	6550	.0	.1	0*	21.1	29.7	71
Basin wide percent of average				29			76
WEBER-OGDEN RIVERS							
TRIAL LAKE	9960	9.9	17.3	57	24.4	30.6	80
THAYNES CANYON	9200	.0	3.3	0	21.0	27.7	76
CHALK CREEK #1	9100	6.4	15.6	41	25.1	30.0	84
MONTE CRISTO	8960	.7	16.9	4	23.5	32.6	72
DRY BREAD POND	8350	.0	7.6	0	17.9	26.8	67
BEAVER DIVIDE	8280	.0	.8	0*	17.6	20.3	87
HORSE RIDGE	8260	.0	5.9	0	23.0	31.1	74
CHALK CREEK #2	8200	.0	3.4	0	17.4	20.3	86
BEN LOMOND PEAK	8000	.5	20.0	2	38.7	54.6	71
FARMINGTON	8000	4.1	6.0	68	35.1	45.2	78
PARRISH CREEK	7740	.0	-M	*	31.2	-M	*
SMITH & MOREHOUSE	7600	.0	1.8	0	17.9	24.0	75
PARLEY'S SUMMIT	7500	.0	3.0	0	21.5	28.3	76
HARDSCRABBLE	7250	.0	3.3	0	27.1	30.6	89
BEN LOMOND TRAIL	6000	.0	1.2	0*	26.8	37.4	72
Basin wide percent of average				20			77

1. Go back to the National Water & Climate Center snow page at www.wcc.nrcs.usda.gov/water/w_data.html [3].
2. This time choose the *Snowpack Maps* link.
3. Select a state. For example, click the *Alaska* link.
4. Choose the *January 2000* button. Your snow map should look similar to the one below.



1. Using your browser, backtrack to the main National Water & Climate Center snow page at www.wcc.nrcs.usda.gov/water/w_data.html [3].
2. This time, click on the *Areal Extent of Snow Cover Maps* link.
3. Select a month and year. For example, choose *January 2000*. Your image should look similar to the one below.



References

[1] http://www.wcc.nrcs.usda.gov

[2] http://www.wcc.nrcs.usda.gov/wcc.html

[3] http://www.wcc.nrcs.usda.gov/water/w_data.html

7.2.c.5. Oceanic Data

GSDA:Oceanic Data

Coastline Data

Obtain coastline data from NOAA NGDC [1]

The National Oceanic and Atmospheric Association (NOAA) has produced the National Geophysical Data Center (NGDC) website to allow for US data access. It is possible to download coastline data from an interactive global map.

- Need help obtaining coastline data from NOAA NGDC, click here.

Tidal Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png [13]

NOAA's National Ocean Service (NOS) is home to the Center for Operational Oceanographic Products and Services (CO-OPS). This center gathers oceanographic data along the coast of the United States. Data available from this site includes historical and real-time observations and predictions of water levels and currents.

Obtain tidal data from NOAA's NOS [2]

Obtain historical tidal data from NOAA's NOS [3]

Obtain tidal precictions from NOAA's NOS [4]

Current Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png [13]

NOAA's National Ocean Service (NOS) is home to the Center for Operational Oceanographic Products and Services (CO-OPS). This center gathers oceanographic data along the coast of the United States. Data available from this site includes historical and real-time observations and predictions of water levels and currents.

Obtain current data from NOAA's NOS [5]

Help with NOAA's NOS [6]

About NOAA's NOS [7]

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_DNR.png [8]

The Texas Coastal Ocean Observation Network (TCOON) monitors several buoys off the coast of Texas. Real-time and historical data can be obtained from this sight. A variety of data types are available for each buoy. More help and information can be found on the web page.

Obtain current data from TCOON [9]

Wave Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_ERDC.png [1]

The Wave Information Studies (WIS) is part of the U.S. Army Corps of Engineers Coastal and Hydraulics Laboratory. Bulk wave parameters (significant wave height, period, direction) and wind speed and direction can be downloaded and viewed on the WIS site.

Obtain wave data from WIS [10]

Help with WIS [11]

About WIS [11]

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NOAA.png [13]

NOAA's National Data Buoy Center (NDBC) supplies data for moored buoys. The buoys measure a variety of data including: barometric pressure; wind direction, speed, and gust; air and sea temperature; wave energy spectra; and direction of wave propagation. Significant wave height, dominant wave period, and average wave period are derived from the wave energy spectra. The data available depends on the buoy type and data provider.

Obtain wave data from NDBC [12]

Help with NOAA's NDBC [13]

About NOAA's NDBC [14]

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_CDIP.png [15]

The Coastal Data Information Program (CDIP) measures coastal environment data. A variety of data types and formats are available on their website.

Obtain wave data from CDIP [15]

Help with CDIP [16]

About CDIP [16]

Oceanic Data Overview

Oceanic Tips

Coastline Tips

- How do I obtain coastline data from NOAA NGDC?
- How do I import coastline data into SMS?

Tidal Tips

- Help with NOAA's NOS [6]
- About NOAA's NOS [7]

References

[1] http://www.ngdc.noaa.gov/mgg/shorelines/shorelines.html
[2] http://egisws01.nos.noaa.gov/website/co-ops/stations/viewer.htm?ActiveLayer=0
[3] http://egisws01.nos.noaa.gov/website/co-ops/stations/viewer.htm?ActiveLayer=3
[4] http://egisws01.nos.noaa.gov/website/co-ops/stations/viewer.htm?ActiveLayer=1
[5] http://egisws01.nos.noaa.gov/website/co-ops/stations/viewer.htm?ActiveLayer=4
[6] http://egisws01.nos.noaa.gov/website/co-ops/stations/helppage.html
[7] http://tidesandcurrents.noaa.gov/about.html

[8] http://lighthouse.tamucc.edu/
[9] http://lighthouse.tamucc.edu/TCOON/HomePage
[10] http://frf.usace.army.mil/cgi-bin/wis/atl/atl_main.html
[11] http://frf.usace.army.mil/wis/WIShowTo.html
[12] http://www.ndbc.noaa.gov/
[13] http://www.ndbc.noaa.gov/faq.shtml
[14] http://www.ndbc.noaa.gov/about_ndbc.shtml
[15] http://cdip.ucsd.edu/
[16] http://cdip.ucsd.edu/?nav=documents&sub=index

GSDA:Obtaining Coastline Data from NOAA NGDC

1. Select the, "Obtain coastline data from NOAA NGDC" link, or go directly with this link http://www.ngdc.noaa.gov/mgg/shorelines/shorelines.html.This will take you to the NGDC shoreline/coastline databases webpage.
2. Click the **Coastline Extractor** link in the middle of the page. It will take you to the extractor tool.
3. Input the bounding coordinates of the coast you wish to get data points for. If you don't know the correct coordinates use the Java Map. Another helpful map is the NGDC Coastal Relief Model.
   As an example use the following coordinates, 121.5° to 123.5° West and 46.5° to 49°North. These correspond to Puget Sound in Oregon.
4. Once you have specified an area you must now select a database. Default database is set at 1:250K. Since Puget Sound is located in the USA choose the *NOAA/NOS Medium Resolution Coastline* in the **Coastline data base** dropdown box. This is the best resolution at 1:70K scale.
   NOTE: There are three more coastline databases which include world data, but range from 1:250K scale at best to 1:5M scale. The 1:70K scale dataset is only available for US coastal data.
5. Defaults for compression, format, and preview should be: none, Mapgen, and GMT Plot respectively. Leave them defaulted and click the **SUBMIT - Extract the Coastline File** button. This will open up a page displaying your requested data and a link to the equivalent data file.
6. Right click on the data link and select **Save Target As**. Then save the file to your computer.
   HINT: Select a more proper name when you save it, like pugetsound. It is possible to save the file as *.DAT or as *.txt; either will be readily assimilated into SMS.

The coastline data that you have sought is now yours. It is possible to import it into SMS models.

7.2.c.6. Surface Characteristics

GSDA:Surface Characteristics

Land Use

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_webGIS.png [5] Land use data from WebGIS [5]

WebGIS offers land use/cover shapefiles in Geographic and UTM coordinates at no cost.

- Additional help obtaining land use data from WebGIS
- View a text file containing the Land Use / Land Cover codes and their definitions
- Help with creating a land use table for Curve Number computation within WMS

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_EPA.png [1] Obtain the HUC number for your watershed [2]

The EPA provides the "Locate Your Watershed" site to help users determine their region's HUC.

Obtain the land use data from the EPA HUC index [3]

The EPA offers land use shapefiles.

- Additional help obtaining land use data from the EPA
- BASINS Metadata website [4]

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_USGS.png [3] Obtain land use data from the USGS [5]

The USGS offers land use data in GIRAS format for ARC/INFO.

- Additional help obtaining land use data from the USGS?
- More information about USGS land use data. [6]

Soil Type Data

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_EPA.png [2] Obtain the HUC number for your watershed [2] Obtain land use and soil type data from the EPA HUC index [3]

The EPA offers land use and soil type shapefiles.

- Additional help obtaining soil type data from the EPA?
- BASINS Metadata website. [7]

http://wikis.aquaveo.com/xms/FTPImages/GSDA/GSDA_NRCS.png [8] Obtain STATSGO soil type data from the NRCS [8]

The United States Department of Agriculture (USDA) supports the Natural Resources Conservation Service (NRCS). A website they have developed is the National Cartography and Geospatial Center (NCGC). Supplied on this website are numerous links and descriptions of geospatial datasets. Through these links it is possible to acquire soil type data maps from the two main soils databases; STATSGO, and SSURGO.

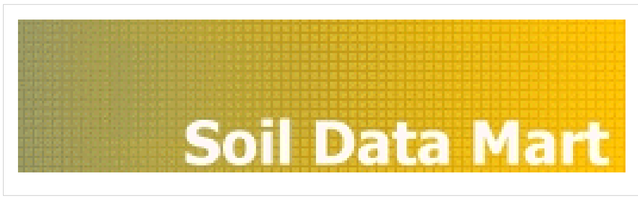
State Soil Geographic (STATSGO) Database

Details:

- Database product files in three formats:

    1. USGS Digital Line Graphs (DLG-3)
    2. ArcInfo 7.0 Coverage
    3. GRASS 4.13 Vector
  - Coordinate systems are:
    1. Albers Conical Equal - Area: Continental U.S. and Alaska
    2. UTM Zone 4: Hawaii
    3. UTM Zone 19: Puerto Rico
  - Resolution at 1:250K scale, except Alaska (1:1M).
  - Complete coverage of U.S. and includes Puerto Rico.
  - Data files are downloadable in zip file compression.
  - Downloads are free (no cost).
- Additional help obtaining soil type data (STATSGO format) from the NRCS

It is possible to obtain SSURGO data from the NRCS by two methods. The first is the Soil Data Mart. The other is through the Geospatial Data Gateway. Both are accessible on the NCGC website.



Obtain SSURGO soil type data from Soil Data Mart [11]

Soil Data Mart is a data request site supported by the United Sates Department of Agriculture (USDA) through the Natural Resources Conservation Service (NRCS). From Soil Data Mart you can place orders requesting certain soil survey maps. It specifically caters to only soil data needs.

Details:

  - Database products available in three forms:
    1. Tabular
    2. Spatial
    3. Template Database
  - Spatial products downloads provided in three formats:
    1. ArcView Shapefile
    2. ArcInfo Coverage
    3. ArcInfo Interchange
  - Coordinate systems are available in a variety of projections:
    1. Geographic
    2. Universal Transverse Mercator (UTM)
    3. State Plane
  - Typical scale resolution between 1:16K and 1:64K.
  - Files can be downloaded only one survey at a time.
  - Data files are acquired in zip file compression.
  - Downloads are free (no cost).
- Additional help obtaining soil type data (SSURGO format) from Soil Data Mart

Obtain SSURGO soil type data from the Geospatial Data Gateway [9]

The Geospatial Data Gateway is a site sponsored by the United States Department of Agriculture (USDA) through the Natural Resources Conservation Service (NRCS). Numerous databases are available by means of the Geospatial Data Gateway including; transportation, census, land use / land cover, soil type, ortho imagery, elevation, and topographic maps. Description below applies to SSURGO soil type data only.

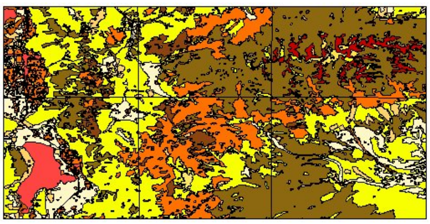
Details:

- Database product files in three formats:
  1. ESRI Shape File
  2. ESRI Coverage
  3. ESRI ASCII Export
- Coordinate systems are available in a variety of projections:
  1. Geographic
  2. Universal Transverse Mercator (UTM)
  3. State Plane
- Typical scale resolution between 1:16K and 1:64K.
- Extensive coverage of U.S. (scheduled for completion in 2008).
- Data files are downloadable in zip file compression.
- Downloads are free (no cost).

- Additional help obtaining soil type data (SSURGO format) from the Geospatial Data Gateway

Land Use Overview

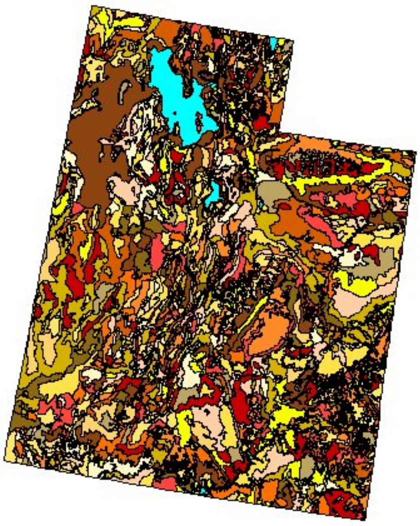
Of the many methods to estimate infiltration, the NRCS Curve Number (CN) method is one that is commonly used. For watersheds with multiple soil types and land uses, a composite CN (CCN) must be calculated to estimate the infiltration losses. The USDA/NRCS supplies tables so that a CCN can be determined from soil type, land use, moisture condition, and hydrologic condition. In hydrology, land use (also known as cover crop or land cover) refers to the way in which the land is being used and/or its condition. Land uses can be urban or agricultural/rural. Examples of these include streets, industrial areas, commercial areas, row crops, meadows, pasture/range and woods.



Land use data for the Provo, Utah region, displayed in ArcView.

Soil Type Overview

For watersheds with multiple soil types and land uses, a composite CN (CCN) must be calculated to estimate the infiltration losses. The NRCS (part of the USDA) supplies tables so that a CCN can be determined from soil type, land use, moisture condition, and hydrologic condition. In hydrology, soil type can be classified in many ways. The USDA classifies them according to their infiltration rate and are referred to as either A, B, C, or D soils. Soil type A has a high infiltration rate whereas soil type D usually consists of clays that are nearly impermeable (low infiltration) and produce higher volumes of runoff.



NRCS STATSGO soil type for the state of Utah.

EPA soil type for an EPA watershed near Provo, Utah.

Land Use Tips

- How do I obtain land use data from the EPA?
- How do I obtain land use data from the USGS?
- How do I format land use data (coordinate system, clipping) for use in WMS?
- How do I decompress data files?
- How do I import EPA land use data into WMS?
- How do I create a land use table for composite CN calculations?

Soil Type Tips

- How do I obtain soil type data from the EPA?
- How do I obtain STATSGO soil type data from the NRCS?
- How do I obtain SSURGO soil type data from Soil Data Mart?
- How do I obtain SSURGO soil type data from the Geospatial Data Gateway?
- How do I format soil type data (coordinate system, soil attribute, clipping) for use in WMS?
- How do I decompress data files?
- How do I import EPA soil type data into WMS?
- How do I create a land use table for composite CN calculations?
- How do I create a soil type runoff coefficient table for composite RC calculations?
- How do I obtain soil type data from the NRCS - STATSGO?
- How do I obtain soil type data from the NRCS - SSURGO?
- How do I use soil type data with GSSHA?

References

[1] http://www.epa.gov/
[2] http://cfpub.epa.gov/surf/locate/index.cfm
[3] http://www.epa.gov/waterscience/ftp/basins/gis_data/huc/
[4] http://www.epa.gov/waterscience/BASINS/metadata.htm
[5] http://edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html
[6] http://edcwww.cr.usgs.gov/glis/hyper/guide/1_250_lulc
[7] http://www.epa.gov/ostwater/BASINS/metadata.htm
[8] http://www.ncgc.nrcs.usda.gov/
[9] http://datagateway.nrcs.usda.gov/

GSDA:Creating a Table of Runoff Coefficients

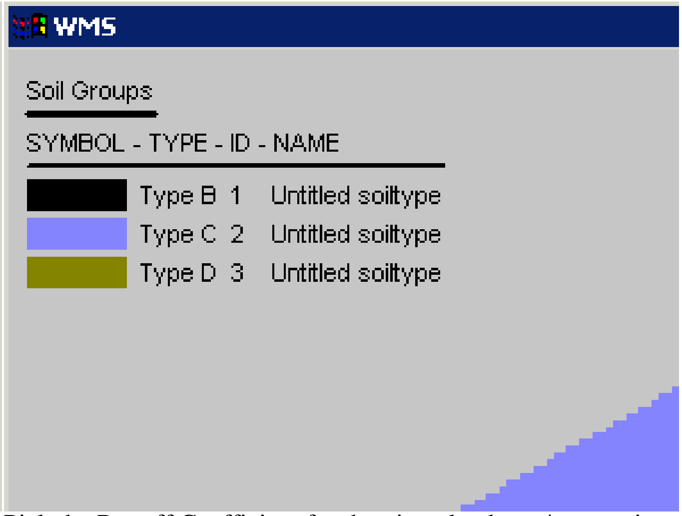
WMS can compute Composite Runoff Coefficients for the basin(s) in your watershed. To accomplish this, you will need to create a table that links a soil type's ID number (from a soil type grid or polygon coverage) to a runoff coefficient. Note that WMS automatically creates a soil ID number for each different soil encountered upon importing a soil type grid or soil type shapefile. Refer to the following steps for help to create a Runoff Coefficient table. Below these steps is a table containing suggested runoff coefficients based on a soil's management type and hydrologic soil group.

Steps for selecting a runoff coefficient for a soil ID number:

For each soil type ID in WMS

1. Determine the land use / vegetation category for the soil ID in question, such as cultivated, pasture, urban, etc.
2. Determine the hydrologic soil group of this ID,
   To determine the soil group (A, B, C, or D) of a soil ID after importing the soil type file into WMS:
   1. Select Display | Display Options...
   2. Choose the MAP tab
   3. Switch the Active Coverage drop-down list to Soil Type
   4. Toggle On the box for displaying the Soil Group Legend
   5. Select OK

   You will now see a small legend in the upper-left corner of the WMS graphics window that lists the soil IDs with their respective soil groups.



3. Pick the Runoff Coefficient for the given land use / vegetation and hydrologic soil group using table like the one below these steps.
4. Create a 3-column text file containing the soil ID, a description, and its coefficient, following this format: soil ID, description (in quotes), runoff coefficient

```
Example

                      0, "Type1", 0.1
                      1, "Type2", 0.2
                      2, "Type3", 0.3
                      3, "Type4", 0.4
                      4, "Type5", 0.5
                      5, "Type6", 0.6
                      6, "Type7", 0.7
                      7, "Type8", 0.8
                      8, "Type9", 0.9
```

6. Save the file, preferably with a *.tbl extension.

This file can now be read into WMS in order to calculate a composite Runoff Coefficient for you watershed basin(s). For help in performing this calculation in WMS, refer to the WMS Help menu.

Table of Runoff Coefficients for the Rational Equation: Q=kCiA

	Hydrologic Soil Group			
Land Use, Crop, and Management	A	B	C	D
CULTIVATED, with crop rotations				
Row Crops, poor management	0.55	0.65	0.70	0.75
Row Crops, conservation mgmt	0.50	0.55	0.65	0.70
Small Grains, poor management	0.35	.40	0.45	0.50
Small Grains, conservation mgmt	0.20	.022	0.25	0.30
Meadow	0.30	0.35	0.40	0.45

```
PASTURE, permanent w/moderate grazing      0.10      0.20      0.25
      0.30
WOODS, permanent, mature, no grazing      0.06      0.13      0.16
      0.20
Urban Residential
30 percent of area impervious            0.30      0.40      0.45
0.50
70 percent of area impervious            0.50      0.60      0.70
0.80

Hydrologic Soil Group Descriptions
A -- Well-drained sand and gravel; high permeability.
B -- Moderate to well-drained; moderately fine to moderately coarse
texture; moderate permeability.
C -- Poor to moderately well-drained; moderately fine to fine texture;
slow permeability.
D -- Poorly drained, clay soils with high swelling potential, permanent
 high water table, claypan, or shallow soils over nearly impervious
layer(s).
```

- This table was obtained from http://pasture.ecn.purdue.edu/~engelb/abe526/Runoff/C_table.html

GSDA:Land Use Codes

```
USGS Land Use and Land Cover Classification System
(see http://landcover.usgs.gov/pdf/anderson.pdf)


1 Urban or Built-up Land

11 Residential
12 Commercial and Services
13 Industrial
14 Transportation, Communications, and Utilities
15 Industrial and Commercial Complexes
16 Mixed Urban or Built-up Land
17 Other Urban or Built-up Land

2 Agricultural Land

21 Cropland and Pasture
22 Orchards, Groves, Vineyards, Nurseries, and Ornamental Horticultural Areas
23 Confined Feeding Operations
24 Other Agricultural Land

3 Rangeland
```

```
31 Herbaceous Rangeland
32 Shrub and Brush Rangeland
33 Mixed Rangeland

4 Forest Land

41 Deciduous Forest Land
42 Evergreen Forest Land
43 Mixed Forest Land

5 Water

51 Streams and Canals
52 Lakes
53 Reservoirs
54 Bays and Estuaries

6 Wetland

61 Forested Wetland
62 Nonforested Wetland

7 Barren Land

71 Dry Salt Flats.
72 Beaches
73 Sandy Areas other than Beaches
74 Bare Exposed Rock
75 Strip Mines Quarries, and Gravel Pits
76 Transitional Areas
77 Mixed Barren Land

8 Tundra

81 Shrub and Brush Tundra
82 Herbaceous Tundra
83 Bare Ground Tundra
84 Wet Tundra
85 Mixed Tundra

9 Perennial Snow or Ice

91 Perennial Snowfields
92 Glaciers
```

GSDA:Obtaining Land Use Data from the EPA

One advantage of the EPA's data is that the land use and soil type data are included together in one download (see Obtaining Soil Type Data).

First, locate the HUC number for your watershed area from the EPA's Locate Your Watershed [2] website. **Second**, go to the EPA BASINS Data HUC Index [1] web site. If the link does not work, follow the bulletted steps below:

- Go to www.epa.gov [2].
- Click on the Information Sources link at the left.
- Click on the Databases and Software link.
- Click on the Geographic Information Systems link.
- Click on the Better Assessment Science Integrating Point and Nonpoint Sources (BASINS) link.
- Click the Web Download link.
- Scroll down and choose the http://www.epa.gov/waterscience/ftp/basins/gis_data/huc link, found under the heading "Basins 3 Data from the Web."

1. Go to the Locate Your Watershed [2] website.

   The website offers four methods of searching for your area's HUC:

   - Search by Map...click on a map of the United States to locate the area
   - Find place...perform a quick search based on a single keyword.
   - Places...use the USGS GNIS website to find your area.
   - Locate by geographic unit...enter the area's zip code, state, county, tribal nation, watershed number or stream name.

   This example will use the Search by Map method.
2. Click on the Search by Map link.
3. Click on a state.
4. Click on the EPA hydrologic unit that covers your watershed.

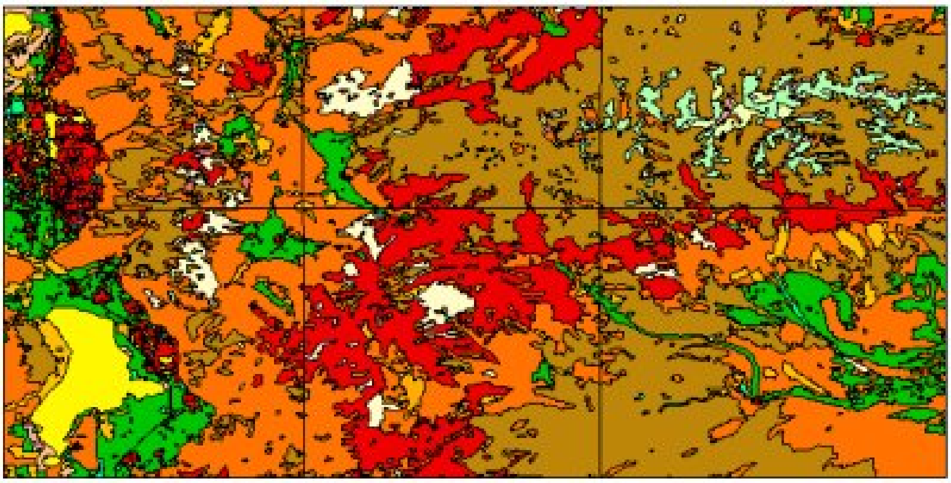
   You may need to download more than one EPA hydrologic unit to completely cover your area. If you find it difficult to ascertain which unit covers your watershed, try using another another search method, such as Find place, for example.
5. Make a note of the 8-digit USGS Cataloging Unit number located just under the Watershed Profile heading.

   This is the HUC number.
6. Go to the EPA's HUC Index [1].
7. Locate and click on the folder named with your 8-digit HUC.
8. Click on the file named with your HUC followed by "_core.exe," (Example: 16020203_core.exe) and in the ensuing dialog, save the file to your computer.

   The downloaded file is a self-extracting zip file.
9. Double click on the saved file to open the "Winzip Self-Extractor" dialog.
10. Specify a location on your computer to where you'd like the files extracted, or unzipped, and then click the Unzip button.

Once you've uncompressed the data file, you should have a new folder made up of the HUC number (e.g., 16020203). Inside of this folder you will find a folder entitled "landuse" which contains one or more land use shapefiles for the EPA watershed. If there is more than one shapefile in this folder, you will need to import them one at at time into WMS or ArcView GIS to determine which one is the correct one.

You've now successfully obtained land use data from the EPA. Don't forget to format (if necessary) the land use data before importing them into WMS.



References

[1] http://www.epa.gov/waterscience/ftp/basins/gis_data/huc
[2] http://www.epa.gov

GSDA:Obtaining Land Use Data from the USGS

The *'USGS Land Use and Land Cover (LULC)* data files include information about vegetation, water, and surface features. Some states are done at 1:250,000 scale whereas others are done at 1:100,000. Each land use polygon has a classification code assigned to it which represents its land use (i.e., classification code 21 is "cropland and pasture").

Click on the link to the *USGS Land Use and Land Cover (LULC)* web site at http://edc.usgs.gov/geodata/.If you have trouble with this link, follow the bulletted steps below. Select "LULC" from the top menu. You can select which land use data you need by clicking on a US map or by selecting the state and region from a list.

- Go to www.usgs.gov [2] on the internet.
- Click on the *Mapping* button near the top.
- Click on the *US GeoData* link under the "Download Data from USGS" heading.

Click on the *LULC* button near the top-right of the screen.

1. Click on the *250K FTP via Graphics* link to get land use data from a map,

    There are several ways to get land use data from the USGS; you can get the data by clicking on a US map, or by clicking on a list of states.
2. Click near the location of interest on the map. For example, click on the state of Utah just underneath the Great Salt Lake.
3. Click on the quadrangle that contains Provo, Utah -- the name of the quad is listed on the map as SALT LAKE CITY, UT.
4. Click on the SALT LAKE CITY, UT link to begin downloading the data.
5. Click on the link for "land use.gz".
6. Proceed to download and save the file onto your computer (e.g., the hard drive). Steps may be slightly different depending on which web browser you're using.
7. Uncompress the file - in this case the file is in ".gz" format, but you can use WinZip to uncompress it. Help Decompressing Files

GSDA:Obtaining Land Use Data from WebGIS

Users can download land use shapefiles from WebGIS in either UTM or Geographic coordinates. Refer to the following steps for guidance on obtaining a land use shapefile from this site:

1. Go to the WebGIS [5] website and notice the three links found under the **LULC Data** Heading on the left side of the page,
2. Click on the *Shapefile (Lat/Long)* link to obtain data in geographic coordinates, or the *Shapefile (UTM)* for data in that coordinate system,
3. Click on a state on the map,
4. Click on a county,

   Be aware that the name of the county that your mouse pointer is over can be viewed in the browser's status bar,
5. Click on one of area links in order to download its corresponding shapefile,
6. Choose "Save Link (Target) As..." to save the file to your computer,
7. Unzip the files to your computer.

   You should now have 3 files that define the shapefile.

To import the shapefile into WMS:

- Open WMS
- Select Import from the File menu
- Change the Files of Type drop-down list to "Feature object polygons Shapefile (*.shp),"
- Navigate to the directory where you saved the files,
- Double click on the file with the *.shp ending,
- Select OK to the Import Shapefile Data dialog. Notice that WMS automatically assigned the LU_CODE field of the shapefiles attribute table to the WMS land use coverage attribute.

GSDA:Obtaining Soil Type (SSURGO) Data from Soil Data Mart

The United States Department of Agriculture (USDA) is a governmental agency which supports the preservation of natural resources and environments through the Natural Resources Conservation Service (NRCS). They support the Soil Data Mart which is an order request site for soil data.

Soil type mapping is a major aspect of maintaining natural environments. Soil type data is supplied in two key forms; one of these is the Soil Survey Geographic (SSURGO) Database, and the other is the State Soil Geographic (STATSGO) Database. STATSGO data is defined on a state-wide level, whereas SSURGO is county-wide survey data. The SSURGO data also has a finer resolution than STATSGO.

Currently, SSURGO data is the only soil type that can be obtained from Soil Data Mart. Most of the United States is covered, but not all. The NRCS is still converting the county data into Shapefile format, it is projected to be completed by 2008. STATSGO data is accessible by other USDA NRCS means.

To download SSURGO soil type data from the Soil Data Mart:

1. Go directly to Soil Data Mart at http://soildatamart.nrcs.usda.gov/.Or choose the **obtain SSURGO soil type data from Soil Data Mart** link.
2. Click the **Select State** button. This will take you to a list of states and territories.
3. Select the state of your choice.
4. Click the **Select County** button. This will bring up a list of counties in the specified state.

   NOTE: You may also browse by survey area, but you will do that when you select a county anyway.
5. Select the county of your choice.
6. Push the **Select Survey Area** button. This will open a list of all available surveys for that county.
7. Select the survey of your choice.
8. Click the **Download Data** button. You will be taken to the data format selection page.
9. Select the *Spatial Data Only* option at the top of the page.

   NOTE: You may also choose the spatial format and coordinate system of your choice. However, leave their default values; they should be ArcView Shapefile and UTM NAD 83 respectively. The reason for using the shapefile format is that it can be directly imported into WMS.
10. Enter your e-mail address in the space provided.
11. Push the Submit Request button. A window will pop-up indicating your order's location in the queue and that you will be notified when it is ready via e-mail.
12. Close the window and wait patiently for the e-mail. It should have *Soil Data Mart* export notification in the subject line.
13. Open the e-mail.
14. Click on the download link. It should initialize the download automatically.
15. Save the file to your computer.
16. Decompress the zip file (Help - Decompressing Files)

Now you have the SSURGO soil type data for the area that you requested. It can directly be imported into WMS since it is already in shapefile format.

GSDA:Obtaining Soil Type (SSURGO) Data from the Geospatial Data Gateway

The United States Department of Agriculture (USDA) is a governmental agency which supports the preservation of natural resources and environments through the Natural Resources Conservation Service (NRCS). Their Geospatial Data Gateway is the main link where resource data can be acquired.

Soil type mapping is a major aspect of maintaining natural environments. Soil type data is supplied in two key forms; one of these is the Soil Survey Geographic (SSURGO) Database, and the other is the State Soil Geographic (STATSGO) Database. STATSGO data is defined on a state-wide level, whereas SSURGO is county-wide survey data. The SSURGO data also has a finer resolution than STATSGO.

Currently, SSURGO data is the only soil type that can be obtained through the Geospatial Data Gateway. Most of the United States is covered, but not all. The NRCS is still converting the county data into Shapefile format, it is projected to be completed by 2008. STATSGO data is accessible by other USDA NRCS means.

To download SSURGO soil type data from the Geospatial Data Gateway:

1. Go directly to the gateway at http://datagateway.nrcs.usda.gov/.Or choose the **Obtain SSURGO soil type data from the Geospatial Data Gateway** link.
2. Click the **Get Data** link on the top of the page. This will take you to step one of acquisition and open a map viewer.
3. Zoom in on the desired area by either single clicking or drawing a bounding box around the area you wish to view.

   HINT: Dragging a box around an area will zoom in faster than clicking on a point. The Zoom In tool is located on the right-hand side of the map window and is the default setting for the map.
4. Continue to zoom in on the map until the desired region is displayed.

   NOTE: As you zoom in closer names for counties, cities, interstates become visible; making it easier to locate your area of choice.
5. Select the *Define Order Area* tool located on the right-hand side.
6. Drag a bounding box around the area of interest.

   NOTE: Upon completing the order area box, you will immediately proceed to step two of acquisition, where a list of available products is displayed.
7. Scroll to the bottom of the page and check the box next to the *Soil Survey Geographic (SSURGO 2.1) DB* option.

   NOTE: A description of your chosen products, including size, will be tallied on the bottom of the window.
8. Click the **Continue to Step 3** button in the bottom left corner. You will be taken to step three of acquisition.
9. Open the dropdown list for **Vector Projection** and select the *UTM Zone 12 NAD 83* option or equivalent.

   NOTE: You may choose the projection of your choice, but remember it for later when, and if you need to convert to another. Leave the other fields; **Vector Extent**, and **Vector File Format** defaulted to *Standard Extent*, and *ESRI Shape File* respectively. The reason for using the shapefile format is that it can be directly imported into WMS.
10. Click the **Continue to Step 4** button in the bottom left corner. You will be taken to step four of acquisition.
11. Fill in the required delivery information fields. Make sure the *FTP Download* option is selected as the delivery method.
12. Click the **Continue to Step 5** button in the bottom left corner. You will be taken to step five of acquisition.
13. Review the information on the order confirmation page to make sure all is correct.
14. Click the **Place Order** button in the bottom left corner and wait for the final confirmation page.

NOTE: The final confirmation page will give you an order number and wait period for when you shall receive an e-mail notification that your order is ready to download.

15. Wait patiently for the e-mail. It should have *From The NRCS Data Gateway* in the subject line.
16. Open the e-mail.
17. Click on the download link. It should initialize the download automatically.
18. Save the file to your computer.
19. Decompress the zip file (Help - Decompressing Files)

Now you have the SSURGO soil type data for the area that you requested. It can directly be imported into WMS since it is already in shapefile format.

GSDA:Obtaining Soil Type (STATSGO) Data from the NRCS

The United States Department of Agriculture (USDA) is a governmental agency which supports the preservation of natural resources and environments through the Natural Resources Conservation Service (NRCS).

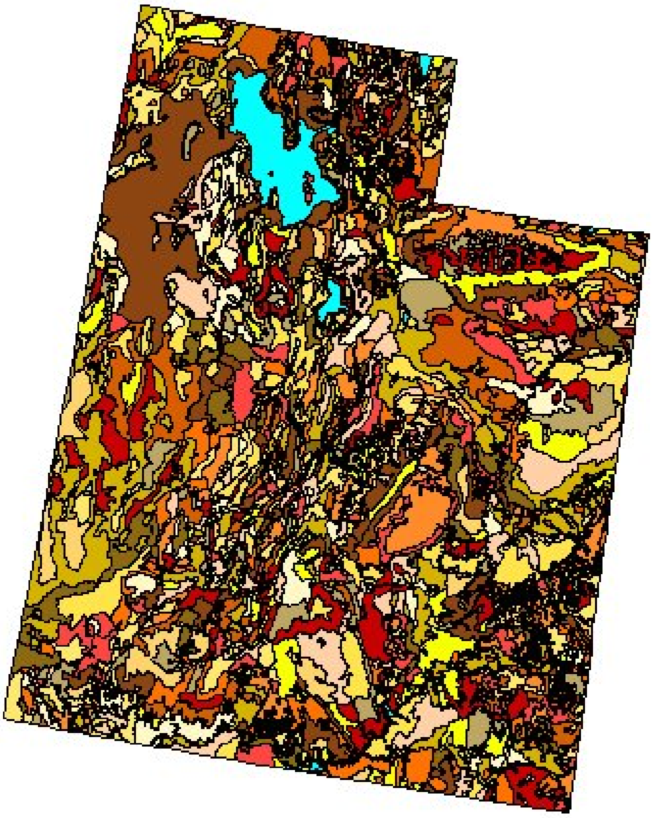
Soil type mapping is a major aspect of maintaining natural environments. Soil type data is supplied in two key forms; one of these is the State Soil Geographic (STATSGO) Database, and the other is the Soil Survey Geographic (SSURGO) Database. STATSGO data is defined on a state-wide level, whereas SSURGO is county-wide survey data.

Currently, STATSGO data is available for the entire United States as well as Puerto Rico. It is possible to access STATSGO information on the National Cartography and Geospatial Center (NCGC) website. This site is managed by the NRCS.

To download STATSGO soil type data from the NRCS:

1. Go to the NCGC homepage at http://www.ncgc.nrcs.usda.gov/.Or choose the **Obtain STATSGO soil type data from the NRCS** link.
2. Click the **STATSGO** link on the left side of the page, under the *Quick Access* heading. This will take you to the STATSGO database home.
3. Click the **Download STATSGO Data** link under the section titled *Database Access*. You will be taken to a list of states.
4. Choose the state you want by clicking on its link. That state's data document downloads will open.
5. Select the **DOS / Windows Data** option below the *ARC / INFO Coverages* header. It should initialize the download immediately.
6. Save the file to your computer.
7. Decompress the zip file. (Tips - Decompressing Files).

Now you have the STATSGO soil type data for the area that you requested. Now you can open up ArcView and import the coverage data. It will be in the folder ...UT / SPATIAL as "UT". Your data should look similar to the picture below.



GSDA:Obtaining Soil Type Data from the NRCS - SSURGO

The NRCS offers two different types of soil type data, namely statsgo and ssurgo. The statsgo data is at a state-wide level, while the ssurgo data is county-wide. The ssurgo data has a finer resolution. The NRCS offers the ssurgo data in a variety of file formats, including DLG(3), Arc/Info Coverage, Arc/Info exchange file (*.e00), and Shapefile (*.shp). For WMS users, the format of choice would be Shapefile, since it can be imported directly into WMS. However, a user owning an Arc/Info or Arcview license can download the Arc/Info Coverage and use Arc/Info or Arcview to convert it to a shapefile.

At the time of composing this page, the NRCS is still converting the county data to the Shapefile format, but is projected to be done in two years or less. As the shapefile data is created, it is posted on the SSURGO website. To date, over 500 counties have the option to download the soil data as a shapefile.

To download soil type data from the NRCS SSURGO website:

1. Go to the SSURGO data web page, found at www.ftw.nrcs.usda.gov/ssur_data.html [1]
2. Click on the Download SSURGO Data link, located towards the bottom of the page.
3. Create a new account if you haven't already done so.
4. Enter your username and password.
5. Click the Download button next to the county of choice.
6. Click the shp.zip button to download the shapefile.
   If the NRCS has not yet compiled a shapefile for your county, you will not see this button.
7. Choose to Save the file to your computer.
8. Unzip the shp.zip file.

9. The shp.zip file contains a folder name "shp." Within this folder you will find several shapefiles; the shapefile containing an "_a" contains the soil type polygon shapes. For example, one might see the following files within the shp directory:

Name	Size	Type	Modified
readme.txt	1 KB	Text Document	2/21/2002 5:05 AM
sc069.gz	838 KB	WinZip File	3/14/2002 12:13 PM
sc069_a.dbf	880 KB	DBF File	2/21/2002 5:05 AM
sc069_a.shp	12,097 KB	AutoCAD Shape So...	2/21/2002 5:05 AM
sc069_a.shx	63 KB	AutoCAD Compiled ...	2/21/2002 5:05 AM
sc069_b.dbf	1 KB	DBF File	2/21/2002 5:05 AM
sc069_b.shp	78 KB	AutoCAD Shape So...	2/21/2002 5:05 AM
sc069_b.shx	1 KB	AutoCAD Compiled ...	2/21/2002 5:05 AM
sc069_q.dbf	5 KB	DBF File	2/21/2002 5:05 AM
sc069_q.shp	3 KB	AutoCAD Shape So...	2/21/2002 5:05 AM
sc069_q.shx	1 KB	AutoCAD Compiled ...	2/21/2002 5:05 AM
schema.ini	1 KB	Configuration Settings	10/22/2002 1:57 PM

10.
11. The file named sc069_a.shp contains the soil type polygons and is the file you would want to import into WMS. Notice that the shp folder contains an additional zip file. This second zip file will have an extension of *.gz and its name will be comprised of the two-letter state abbreviation plus 3-digits (for example, "sc069.gz"). This zip file contains a series of text files (*.txt). These files contains attribute data for the soil type polygon shapefile. Continue with the following steps:
12. Unzip the attribute data zip file.
    Note: The file named comp.txt contains the Hydrologic Soil Group field, which you will need to join to the soil type polygon shapefile.
    To join the Hydrologic Soil Groups to their respective soil type polygons, you can either use the Excel Template provided below, or use a GIS Application, like Arcview.
    Help – performing a Join with Arcview

The ssurgoImport.xls Template:

This spreadsheet template contains macros that are designed to:

- Quickly import the comp.txt file, which contains the soil attribute data for the polygon shapefile
- Join its Hydrologic Soil Group field to the shapefile attribute table
- Save out the new shapefile attribute table containing the Hydrologic Soil Group (A,B,C, or D) for each polygon record

This template was especially designed for WMS (Watershed Modeling System) users who are obtaining soil type shapefile data from the NRCS SSURGO website.

The NRCS also provides a Microsoft Access template for reading the text file data located in the attribute data zip file. This template is more robust for more general GIS applications and enables users to quickly produce reports from the attribute data. This template, along with information on its use, can be obtained from the NASIS (National Soil Information System) web site at http://nasis.nrcs.usda.gov/downloads/.

Download the Microsoft Excel Template [2]

Note:

- This template contains macro programs. In order to function, you will need to make sure that Excel's security level is set to medium before opening the template. This can be done by selecting *Macro* from the *Tools* menu, then selecting *Security...*

- This template is designed to work in Excel 2003. To be able to use it in Excel 2007, you will need install an excel add-on that allows you to save *.dbf files

References

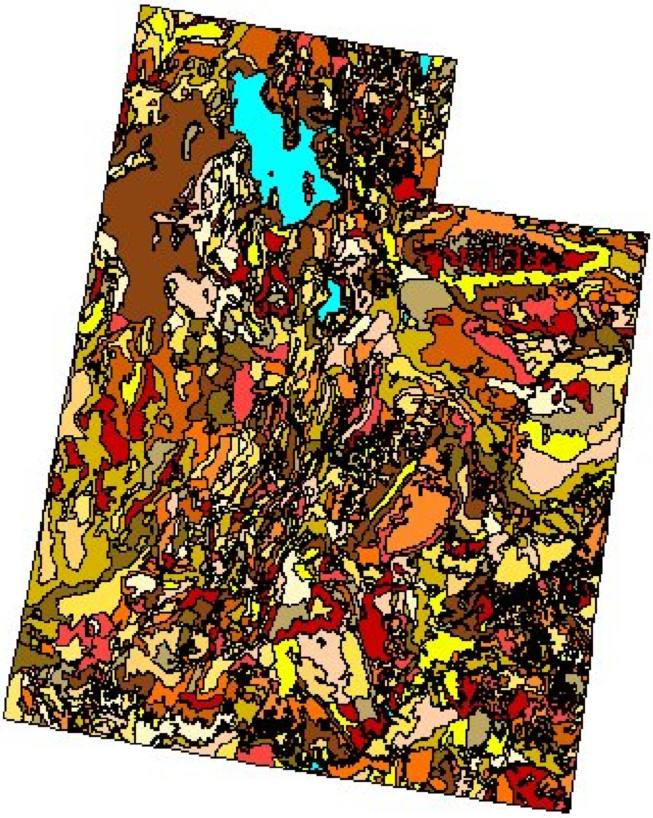
[1] http://www.ftw.nrcs.usda.gov/ssur_data.html
[2] http://wms.aquaveo.com/ssurgo.zip

GSDA:Obtaining Soil Type Data from the NRCS - STATSGO

The NRCS offers statewide soil data in Arc/Info coverage format, which must be converted to a shapefile before the soil data can be imported into WMS. Go to the NRCS soil type web site at www.ftw.nrcs.usda.gov/stat_data.html [1]. If you have trouble getting there, follow the bulletted steps below:

- Go to the USDA web site at www.usda.gov [2].
- Click on the **Agencies & Offices** button near the top.
- Scroll down and select the **Natural Resources Conservation Service (NRCS)** link near the bottom of the *USDA Agencies* section.
- Click on the **Technical Resources** link under the Programs and Service Highlights heading. A new window will open on the Technical Resources page of the NRCS home.
- Choose the **Soils** link.
- Choose the **STATSGO** link on the left-hand side of the page, under the *Quick Access* section.
- Select the **Download STATSGO Data** link under the section titled *Database Access*. You will be taken to a list of states.
- Choose the state you want by clicking on its link. That state's data document downloads will open.
- Select the **DOS / Windows Data** option below the *ARC / INFO Coverages* header. It should initialize the download immediately.
- Save the file to your computer.
- Decompress the zip file (Help - decompressing files)

1. Choose the *Forms-based Download* link to obtain data.
2. Click the *DOS/WINDOWS* link and choose the "/arc" folder.
3. Now click the "/data" folder.
4. Finally, do a single right-click on the state abbreviation for which you need data. For this example choose Utah (ut.zip). Then choose "Save Link As..." or "Save Target As..." depending on whether you use Netscape Navigator or Microsoft Explorer respectively. Proceed to download the data.
5. Uncompress the data. For help, see Decompressing Files.
6. Once in ArcView, the ARC/INFO coverage data can be found in the "Ut\Spatial" as "Ut". Your data should look similar to the picture below.



References

[1] http://www.ftw.nrcs.usda.gov/stat_data.html

GSDA:Table of Runoff Curve Numbers

Description of Land Use	**Hydrologic Soil Group**			
	A	**B**	**C**	**D**
Paved parking lots, roofs, driveways	98	98	98	98
Streets and Roads:				
Paved with curbs and storm sewers	98	98	98	98
Gravel	76	85	89	91
Dirt	72	82	87	89
Cultivated (Agricultural Crop) Land*:				
Without conservation treatment (no terraces)	72	81	88	91
With conservation treatment (terraces, contours)	62	71	78	81
Pasture or Range Land:				
Poor (<50% ground cover or heavily grazed)	68	79	86	89
Good (50-75% ground cover; not heavily grazed)	39	61	74	80
Meadow (grass, no grazing, mowed for hay)	30	58	71	78
Brush (good, >75% ground cover)	30	48	65	73
Woods and Forests:				
Poor (small trees/brush destroyed by over-grazing or burning)	45	66	77	83
Fair (grazing but not burned; some brush)	36	60	73	79
Good (no grazing; brush covers ground)	30	55	70	77
Open Spaces (lawns, parks, golf courses, cemeteries, etc.):				
Fair (grass covers 50-75% of area)	49	69	79	84
Good (grass covers >75% of area)	39	61	74	80
Commercial and Business Districts (85% impervious)	89	92	94	95
Industrial Districts (72% impervious)	81	88	91	93
Residential Areas:				
1/8 Acre lots, about 65% impervious	77	85	90	92
1/4 Acre lots, about 38% impervious	61	75	83	87
1/2 Acre lots, about 25% impervious	54	70	80	85
1 Acre lots, about 20% impervious	51	68	79	84

from Chow et al. (1988)

GSDA:Using Soil Type Data with GSSHA

This website [1] has more information.

SSURGO GSSHA Soil Data Preparation Spreadsheet

Click here [2] to download SSURGO GSSHA soil data preparation spreadsheet

References

[1] http://gsshawiki.com/index.php?title=Mapping:Assigning_spatially_distributed_parameters
[2] http://wms.aquaveo.com/GsshaSsurgoImport.zip

7.2.c.6.1. Obtaining Soil Type Data from the EPA

GSDA:Obtaining Soil Type Data from the EPA

The EPA offers STATSGO soil type data in one large "core data" download. One advantage of the EPA's data is that the land use and soil type data are included together in one download. The steps for obtaining the soil data are outlined below.

First, locate the HUC number for your watershed area from the EPA's Locate Your Watershed [2] website.

Second, go to the EPA BASINS Data HUC Index [1] web site. If the link does not work, follow the bulletted steps below:

- Go to www.epa.gov [2].
- Click on the *Information Sources* link at the left.
- Click on the *Databases and Software* link.
- Click on the *Geographic Information Systems* link.
- Click on the *Better Assessment Science Integrating Point and Nonpoint Sources (BASINS)* link.
- Click the *Web Download* link.
- Scroll down and choose the http://www.epa.gov/waterscience/ftp/basins/gis_data/huc" *link, found under the heading "Basins 3 Data from the Web."*

1. Go to the Locate Your Watershed [2] website,

   The website offers four methods of searching for your area's HUC:

   - Search by Map...click on a map of the United States to locate the area
   - Find place...perform a quick search based on a single keyword.
   - Places...use the USGS GNIS website to find your area.
   - Locate by geographic unit...enter the area's zip code, state, county, tribal nation, watershed number or stream name.

   This example will use the Search by Map method,
2. Click on the Search by Map link,
3. Click on a state,
4. Click on the EPA hydrologic unit that covers your watershed,

   You may need to download more than one EPA hydrologic unit to completely cover your area. If you find it difficult to ascertain which unit covers your watershed, try using another another search method, such as Find place, for example.
5. Make a note of the 8-digit USGS Cataloging Unit number located just under the Watershed Profile heading,

   This is the HUC number.
6. Go to the EPA's HUC Index [1],
7. Locate and click on the folder named with your 8-digit HUC,
8. Click on the file named with your HUC followed by "_core.exe," (Example: 16020203_core.exe) and in the ensuing dialog, save the file to your computer,

   The downloaded file is a self-extracting zip file.
9. Double click on the saved file to open the "Winzip Self-Extractor" dialog.

10. Specify a location on your computer to where you'd like the files extracted, or unzipped, and then click the Unzip button.

Once you've uncompressed the data file, you should have a new folder made up of the HUC number (e.g., 16020203). Inside of this folder you will find a file named "statsgo.shp." This is the primary soil type shapefile containing the polygon boundaries for the soil types of your area. Scrolling down the list of files, you will also find a file named "statsgoc.dbf." This file contains information on the Hydrologic Soil Group and Surface Textures.

For computing composite Curve Number and/or Runoff Coefficient calculations with WMS, you will need to join the Hydrologic Soil Group field, hydgrp, found in the statsgoc.dbf file to the statsgo.shp file using the spreadsheet template provided below, or a GIS application such as ArcView®.

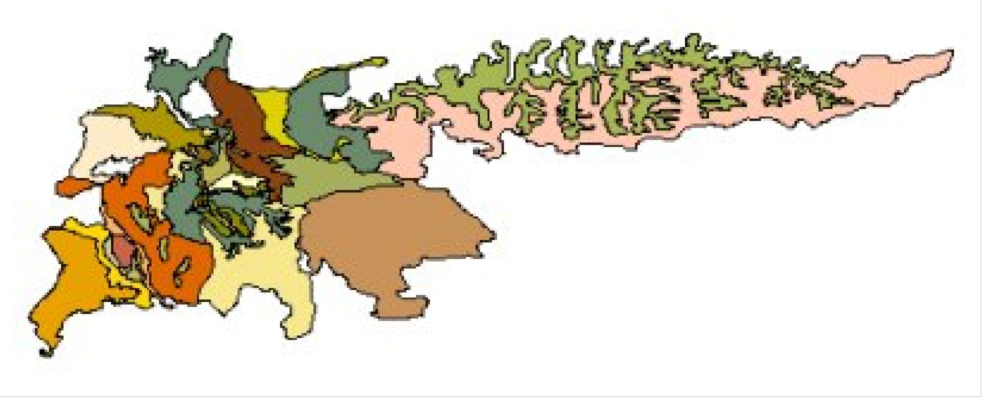
statsgoImport.xls Template

This template contains a VBA macros designed to open both the statsgo and statsgoc tables into a spreadsheet. Then with a single click, users can join the hydgrp field from the statsgoc table to corresponding records in the statsgo table.

Click here to download the statsgoImport zip file, which contains the template and its readme file. [1]

Be sure to view the readme.txt file for instructions on using the template.

Brief description of how to join the *hydgrp* field to the soil type shapefile using ArcView GIS (3.x).



References

[1] ftp://pubftp.ems-i.com/download/WMS/misc/statsgoImport.zip

GSDA:StatsgoImport readme

StatsgoImport readme

```
This Excel template allows you to open the statsgo.dbf and statsgoc.dbf
 files in an Excel spreadsheet, and then join the HYDGRP field of the
statsgoc table to the statsgo shapefile table.  Both the statsgo and
statsgoc tables are part of the EPA Soil Type core data download.

Instructions for using statsgoImport.xls

1) Unzip the statsgoImport.zip file

2) Open Microsoft Excel

3) Open statsgoImport.xls

4) Select Enable macros

5) Select "Yes" to open the worksheet as Read-Only
   This will ensure that the original template does not get changed
   i. e. to make changes, you will have to save the file with a new
name.

--- To import the statsgo and statsgoc files

1) On the gray form, select the Open statsgo.dbf button

2) Browse for and open the statsgo.dbf file

3) Select the Open statsgoc.dbf button

4) Open the statsgoc.dbf file

--- To join the statsgoc HydGrp field to the statsgo attribute table

1) Click the Join button
       This command first sorts the statsgoc table in descending order
       based on its COMPPCT (composition percentage) field.  If
multiple Map Unit Key ID's exist within the statsgoc table, then
sorting the data by comppct ensures that the Join function selects the
HydGrp attribute for the largest contributng area of the map unit.

2) Click the Fill button to make sure all records have a HydGrp entry
          Enter a Hydrologic Soil Group (A,B,C, or D) or other string to
 use as the "filler."
```

```
5) Click the Clean button to convert any hybrid soil types to a single
type
         For example, if the entry is "B/D," then this command changes
the entry to
         be the first group listed, "B," in this case.  The Composite
Curve Number
         calculator in WMS requires that each soil type polygon have a
single soil type,
         A, B, C, or D, and not mixtures.
         If you do not plan to use the Curve Number calculator, you can
 skip using the
         Clean button, or enter some other text as the filler.

6) Click the Save new dbf button

7)  Enter a filename and click the Save button
          You should overwrite the original *.dbf file in order for it
to remain
          "linked" to the *_a.shp file.

```

```
NOTE:

This file meant as a utility for users who want to download STATSGO
data from the EPA's
Core BASIN data archive and quickly format the the soil type shapefile
for importing it into WMS (Watershed Modeling System).
```

Have any constructive comments on this template? send them to oneil_h@yahoo.com

7.3. Archives

Archived Models

The following models are either obsolete or have not yet been released to the public.

1. Cascade
   - Cascade
2. Dredge Source Model
   - Dredge Source Model
   - Dredging Material Properties_Stickiness
   - Dredging Scheduling
   - Dredging Sediment Characteristics
   - Dredging Simulations
   - Importing Dredge Tracks from ASCII Files
3. FATE
   - FATE
   - FATE Menu
   - FATE Model Control Current
4. GENESIS
   - GENESIS
   - GENESIS Graphical Interface
   - GENESIS Model Control Dialog
   - GENESIS Structures
   - GENESIS Menu
   - GENESIS Coastal Structures
5. HEC-RAS
   - HEC-RAS
   - HEC-RAS Material Properties
   - Q&A HEC-RAS
6. Holland\PBL
   - Holland\PBL
   - Hurricane Path Pertubations Dialog
   - Holland Symetrical/Asymetrical
   - PBL
   - ADCIRC Wind Coverage PBL
7. LTFATE
   - LTFATE
   - LTFATE Menu
8. STFate
   - STFate
   - STFate Cloud File
   - STFATE Clouds to PTM Sources

- STFate Compliance Reports

7.3.a. What's New for Previous Versions

What's New in SMS 11.0

SMS 11.0 includes a new raster module, interfaces for several new numeric engines, as well as new and updated features to make it possible for you to do more things, do things better or do things faster.

Updates

```
Normal  0         false  false  false    EN-US  X-NONE  X-NONE
```

SMS 11.0.12 – Built November 8, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version can patch SMS 11.0.7 through 11.0.11 (earlier versions require a full-install).

SMS 11.1 has now been distributed in beta form for almost two months and a full install is expected soon. Unless critical bugs are identified, this will be that last bug-fix release for version 11.0 of SMS.

Bugfixes

The following bugs have been fixed in this version.

1. Error in converting a stamped coverage to a scatter set was resolved
2. When loading a generic mesh in a *.2dm file, with a template that does not correspond to the current model definition, SMS could crash. This problem was resolved to allow for preservation of the grid.
3. In some situations, when generating a drogue plot or particle trace animation, some of the particles were not getting the new time step data. This has been resolved.
4. Fixed an error related to interpolation to nodes that are “inactive”. All nodes now get interpolated data regardless of the state of the connected elements.
5. Calibration plots using the active data set did not update when changing to a completely different scatter set. This has been resolved.
6. Fixed a drawing order that caused annotation items to appear on top of the film loop clock. The clock now appears on top.
7. Additional comments were added to the TUFLOW Flow Constriction dialog to make the units and coefficients more understandable.
8. Bug related to reading in a large RMA2 solution was resolved.
9. When SMS writes GIS data (or a coverage) as a “*.mif”/”*.mid” pair, an extra “bound” record was being written. This has been corrected.
10. Calibration targets are now not drawn when the observation point falls in an inactive cell/element.
11. When framing the display, SMS now includes active grid frames.
12. Fixed erase behind labels to include the film loop clock.
13. When creating datasets in the dataset toolbox, if all arguments to an operation come from a single folder, the result will now be placed in the same folder.
14. Fixed a situation where the screen froze trying to display contour labels.

SMS 11.0.11 – Built October 5, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version can patch SMS 11.0.7 through 11.0.10 (earlier versions require a full-install).

Bugfixes

The following bugs have been fixed in this version.

1. Several internal memory leaks were corrected.
2. How SMS saves the input wind file for the “Holland Symmetrical” hurricane option for wind in ADCIRC were corrected.
3. An error related to material sets in TUFLOW was corrected. SMS would lose track of which material set was assigned to a simulation when a project was saved and then read back in.
4. In some cases it appeared that SMS was deleting data sets from the Project Explorer. In reality, there were just some empty folders that were left in a simulation.
5. Clarification was made for the “Override Z values” option for TUFLOW in both the dialog and the documention.
6. In the dataset toolbox, when SMS creates a new data set, the activity was being computed from the resultant nodal activity. This caused single cell/element islands that were inactive to become active. This was changed to use the element activity of the input data sets to compute the resulting element activity.
7. The ordering of external files in the geometry component file (tgc) is now controlled by the order of the tree items to allow the user to control this explicity.
8. A crash when loading ADCIRC grid files was resolved. Some files had “Unexpected End of File” messages, and then crashed. They now read in correctly as they did with previous versions.
9. Error in the TUFLOW interface to “Force cell z to at or below the node z” was resolved. The option is now available without activating the “Set Channel Invert” elevation.
10. Inconsistent display of rasters corrected.
11. Error in the SMS steering of CMS-Wave and CMS-Flow was corrected to save the correct eta files.
12. A file pointer to a shape file was not released when SMS opens the file. This has been corrected.
13. A issue with user defined palettes was corrected. The first palette was always active when reloading a project. Now SMS looks for a named palette that was active when the project was saved. If that palette exists, it is made current.
14. SMS has been known to save an illegal material with an ID of zero when saving TUFLOW projects. This should not happen any more.
15. SMS allows two TUFLOW boundary condition coverages. They cannot be identical or TUFLOW will crash.
16. Fixed an issue of editing BC curves associated with arcs in a TUFOW coverage. Changing the curve used to change it for other copies. The distinction of multiple curves is now maintained.
17. Fixed a bug with how WAM grids read in native format are loaded into SMS.
18. Fixed output of GCL strings for RMA4 to not include the midside nodes.

SMS 11.0.10 – Built September 10, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version can patch SMS 11.0.7 through 11.0.9 (earlier versions require a full-install).

Bugfixes

The following bugs have been fixed in this version.

1. Additional documentation was added to the help manual on the AdH model control options.
2. A memory overflow error when triangulating is detected so that SMS does not crash when triangulating very large scatter sets. Several possible solutions to get the data loaded are suggested. A future enhancement will reduce memory requirements for triangulation. This also applies when merging two large scatter sets that cause SMS to try to retriangulate.
3. Fixed problem when loading STWAVE output files and converting the data sets to scalars. The conversion was creating scalar data sets with “0.0” values.
4. Fixed an issue with graphics when dragging an arc in the Map Module. During the drag, a phantom arc would appear offset from the arc that was being dragged.
5. Fixed a problem when forcing breaklines into a TIN that resulted in flat or degenerate triangles. No degenerate triangles should be created now.
6. A problem with RMA4 giving bad values at midside nodes was isolated and reported to the TABS team at ERDC. No update is expected.
7. A problem was resolved in the “Holland Symmetrical” hurricane option (NWS = 8).
8. Resolved an issue with calibration plots when multiple solutions exist.
9. Fixed spacing in the model wrapper when running AdH to match what is output in a command prompt. This makes the output more readable.
10. Fixed a crash when creating a telescoping grid for CMS-Flow.
11. Resolved an issue when reading a TUFLOW solution with overlapping elements at the confluence of two 1D tributaries. SMS deletes the longer (skinnier) triangles that were overlapping, but still reads the data sets for visualization. When these cross sections are properly constructed, there should not be overlapping triangles.
12. Made change so that when user tries to open the HURDAT database, SMS switches to geographic coordinates to support the data if possible.
13. Fixed crash related to SMS trying to switch into PBL model even though the interface is not licensed.
14. Resolved a situation which caused SMS to freeze when dragging a coverage to another location in the project explorer (in Map Module or in a TUFLOW simulation).
15. Clarified the management of linear –vs- quadratic elements in the mesh module. If both types are permissible, SMS now asks the user to specify the desired type. The default is set to quadratic if quadratic elements exist of when working with TABS or FESWMS. Otherwise, the default is linear.
16. Fixed a crash when loading multiple map files into SMS.
17. Added check to verify validity of a coastline file before reading to prevent crash.
18. Fixed a situation where SMS assigns the wrong material type to polygons created during a conversion from a mesh.
19. Added support for “TD Card” (the global time step) when converting SMS 10.1 generic model templates to SMS 11.0 format.
20. Added a warning to let users of the generic model interface be informed that just reading a 10.1 template will not allow use of an 11.0 generic model version.
21. Fixed conversion of air density for FESWMS when switching from customary units to SI.

22. Implemented a change to maintain the material set assignments for TUFLOW simulations for an SMS project when saving and reopening the project.
23. Corrected a linkage problem with scatter data sets being lost when deleting other data sets.
24. Clarified the documentation and prompts when using the “override Z values” options in the TUFLOW interface. The model supports several options that were causing confusion among users.
25. Added a right click command in the 1D-2D connections coverage properties to allow users to select the boundary condition coverage to use with 1D-2D connections in the TUFLOW interface.
26. Fixed a problem with CMS-Flow/CMS-Wave steering that was preventing the “eta” or surge file from being passed to the wave model after a flow simulation.
27. Fixed a problem with the names of rasters being reset to default value after saving.

SMS 11.0.9 – Built August 6, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version can patch SMS 11.0.7 and 11.0.8(earlier versions require a full-install).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed an issue in the CMS-Flow model wrapper to ensure the run time would update.
2. Fixed a problem with the names of rasters being reset to default value after saving and reloading an SMS project.
3. Fixed problem with visualization mode to support quadratic elements. This allows switching between visualization mode and TABS without problems.
4. Fixed error in computing flux across an observation arc. Documented that insufficient resolution, either by saving only TUFLOW corners, or by unrepresented AdH refinement can cause errors in computed flux.
5. Fixed bug with bad air mass density when converting from customary units to SI and back.
6. Fixed bug with node labels that stayed visible when nodes were turned off.
7. Fixed display of node numbers so that they do not overlap nodal elevations when both are turned on.
8. Fixed a problem with the name that was written to the *.mp file for CMS-Flow when extracting both WSE and velocities along a boundary cellstring.
9. Fixed a bug when reading in *.2dm file boundary condition on a deleted node string.
10. Fixed issue with the display of vectors after the mesh is turned off.
11. Fixed a crash in the AdH materials property dialog when using sediments.
12. Fixed a tolerance issue when using the Paving function with geographic coordinates and inserting a new mesh into an existing mesh.
13. Fixed crash when trying to open an STWAVE *.sim file as a CMS Wave *.sim file.
14. Fixed a bug with the TUFLOW 2D Model Control switching from a Coverage selector to an edit field.
15. Fixed a crash that occurred sometimes when exporting TUFLOW files.

SMS 11.0.8 – Built July 3, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version can patch SMS 11.0.7 (earlier versions require a full-install).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed an error with exporting STWAVE simulation files.
2. Fixed a bug that where SMS was crashing when trying to remesh a portion of a mesh when quadratic elements were used.
3. Fixed an issue where nodestring IDs were not read in correctly.
4. Fixed a problem where some image files did not display until another item was read in.
5. Fixed a problem with the generation of damping cellstrings for BOUSS2D.
6. Corrected an issue first seen in SMS 11.0.4 where internal arcs were not handled correctly when creating a 2D mesh.
7. Allowed negative values for TUFLOW output start times since TUFLOW allows negative times.
8. Fixed an issue with ADH iteration controls not reading/writing correctly.
9. Fixed a crash that sometimes happened when converting GIS data to feature objects.
10. Fixed a problem where the initial contour display was a solid blue rather than specified values.
11. Corrected a problem where TUFLOW material sets were put into folders incorrectly.
12. Fixed an issue where SMS was creating bad triangles when merging scattersets.
13. Dragging of feature nodes now displays connecting lines while dragging.
14. Removed an incorrect model check warning when using extracted boundary conditions with CMS-FLOW.
15. Fixed a crash that sometimes happened when opening two *.tcf files at the same time.
16. Various bugfixes from software crash reports.

SMS 11.0.7 – Built May 31, 2012

This is a bug-fix release for SMS 11.0.

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version requires doing a full install (no update patch for this version).

Bugfixes

The following bugs have been fixed in this version.

1. PTM will no longer show not licensed when run from SMS using a single user lock.
2. Mesh quality not displayed when a mesh is turned off in the project explorer.
3. Datasets are preserved when saving/loading a project file which wasn’t always working.
4. When displaying labels on scalar values at mesh nodes, the labels could disappear sometimes. This has been fixed.
5. Fixed a problem where reading certain *.mif/*.mid files crashed SMS.
6. Fixed a problem where SMS was hanging when cleaning/merging arcs.
7. SMS sometimes gave an error message “Unable to write file path for keyword “:NEIGHBOR_FILE” when saving an SMS project.
8. Fixed a problem where SMS would freeze trying to perform zonal classification.
9. The CMS-FLOW folder was reorganized to allow 32 and 64 bit executables to both work.
10. Sometimes SMS was not maintaining the folder structure for loaded XMDF dataset files.

11. Fixed a problem where zonal classification was not working correctly for Cartesian grids.
12. TUFLOW events now prevent commas from being used as they don’t work with the TUFLOW file formats.
13. Fixed an issue where right-clicking on coverage and choosing Map→2D Mesh would operate on the active coverage rather than the one clicked on.
14. Fixed a problem where the TUFLOW files for geometry components were written out incorrectly when multiple components exist.
15. Fixed an update problem when using user defined palettes.
16. Fixed a problem where rotating could cause parts of a functional surface to disappear.
17. Fixed a problem where the data calculator was not correctly using inactive cells for CMS-FLOW.
18. Fixed a location where SMS was giving a prompt that the project had changed when it hadn’t.
19. Changed SMS to not reframe when performing a map→2D mesh.
20. Fixed a crash when working with user defined tidal constituents in ADCIRC.
21. Fixed an issue where breaklines of a scatterset were being displayed even though the scatterset was turned off in the project explorer.
22. Fixed a problem where the runtime wasn’t being updated when running CMS-FLOW.
23. Turned off a bogus model checker when using CGWAVE when using approximated wave conditions.
24. Fixed a series of bugs related to setting up a generic model template.
25. Fixed an issue where SMS was not writing the correct number of frequencies to the STWAVE *.eng file.

SMS 11.0.6 – Built May 4, 2012

This is a bug-fix release for SMS 11.0. This is a special update to address a couple specific issues that were introduced in 11.0.5 (items 1 & 2 below).

We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version includes an update that can be installed on top of SMS 11.0.1 (sorry can’t patch 11.0.0 due to an issue with our installation software).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed a bug that was introduced in SMS 11.0.5 where SMS would hang when trying to build polygons.
2. Included a file needed to check for AVI codecs to make sure they would work with SMS. This file is only needed for versions starting with SMS 11.0.5.
3. Fixed a minor issue where the welcome dialog reported that the SMS was a beta version.
4. Miscellaneous changes based upon crash logs sent from the software (bugtrap).

SMS 11.0.5 – Built April 25, 2012

This is a bug-fix release for SMS 11.0. We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version includes an update that can be installed on top of SMS 11.0.1 (sorry can’t patch 11.0.0 due to an issue with our installation software).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed a bug that caused SMS to freeze when using a large duplicate node tolerance.
2. Corrected the executable used by the BOUSS2D “1D Runup and overtopping calculator.”
3. Fixed a problem where framing and switching to plan view could change the display of elements and nodes.
4. Tightened the tolerance used when loading RMA2 files to prevent curves from losing information when loading.

5. Fixed a problem that sometimes made the generic mesh interface hang.
6. Fixed a problem where the specified image projection was not written out causing the need to respecify the projection everytime the project was loaded.
7. Fixed a problem where portion of scatter set not displayed correctly when not filling above/below contour range.
8. SMS sometimes fouled up a polygon or crashed when moving nodes in the mesh attributes dialog (polygon attribute in mesh coverage).
9. Corrected an issue where the drogue times in the log file were not reported correctly.
10. Miscellaneous changes based upon crash logs sent from the software (bugtrap).

SMS 11.0.4 – Built March 3, 2012

This is a bug-fix release for SMS 11.0. We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version includes an update that can be installed on top of SMS 11.0.1 (sorry can't patch 11.0.0 due to an issue with our installation software).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed a problem where sometimes a plot didn't generate correctly for arcs without interior vertices
2. SMS now closes STWAVE model files after running the model.
3. Fixed a problem that could occur when closing spectral energy plot and then reopening the plot.
4. Fixed an issue with the mesh quality min angle not working correctly.
5. Fixed an issue where the correct tidal constituents was not being displayed when using the Harmonic Analysis option with ADCIRC.
6. Fixed a problem with default values for nodestrings when using the generic mesh model.
7. Fixed a problem where the same coverage name in different folders couldn't be differentiated when using a coverage to display vectors.
8. Fixed a problem where AdH Iteration control parameters were not being saved.
9. Fixed a problem with reprojecting a Cartesian grid incorrectly reporting the minimum data value.
10. Fixed a problem with curve BC data when using the generic mesh model.
11. Fixed a problem where SMS was freezing when converting scatter breaklines to map.
12. STWAVE half-plane model only supports spectral energy at 5 degree direction bins. SMS was allowing users to specify a different angle which led to problems when running the model.
13. SMS was not handling the float/curve option for the generic mesh model correctly.
14. Fixed a problem where SMS gave a projection error message when loading an image even though the projection was valid.
15. Fixed a crash when trying to paste data into the cross-section attributes dialog.
16. Fixed a crash that could sometimes happen when redistributing vertices inside the polygon attributes dialog.
17. Fixed an issue with losing BC information specified using the generic model interface.
18. Fixed a problem where the spectral index wasn't being saved correctly.
19. Fixed a problem in how STWAVE with breaking datasets when using XMDF dataset output.
20. Fixed an issue where SMS incorrectly told the user the STWAVE data had changed after saving a file.
21. Made it so that you can control which raster is the active raster.
22. Fixed a problem with zoom to raster item.
23. Made it so SMS would switch modules if you clicked the root raster item.
24. Add raster module to right-click switch modules in empty area of project explorer.
25. Fixed a problem where SMS could freeze when trying to build polygons.
26. Fixed an incorrect model check when working with the CGWAVE model.

27. Fixed a problem with launching generic mesh models.
28. Fixed a crash that could occur when specify custom tidal constituents for ADCIRC.
29. Fixed a crash that could sometimes happen when using functional surfaces and moving windows.
30. Made it so SMS correctly used the projection associated with a scatterset when reading.
31. Corrected an issue where datasets created in the dataset calculator could end up in the hotstarts datasets when using AdH.
32. Fixed a problem where the meshing preview in the polygon attributes dialog for mesh coverages could mess up the polygon’s arcs.
33. Fixed a crash with PTM and trap files.
34. Fixed a problem where an error message similar to “No dataset specified for interpolation” when using observation plots with scattersets.
35. Fixed a crash that could sometimes happen when specifying global parameters for a generic mesh model.
36. Fixed an error where SMS gave a generic error message when trying to run STWAVE that did not identify the problem.
37. Fixed a problem where incorrect model setup for WAM could lead to a crash when trying to run the model.
38. Fixed a problem where contours were not being displayed for elements that were wet but being treated as dry.

SMS 11.0.3 – Built Jan 27, 2012

This is a bug-fix release for SMS 11.0. We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version includes an update that can be installed on top of SMS 11.0.1 (sorry can’t patch 11.0.0 due to an issue with our installation software).

Bugfixes

The following bugs have been fixed in this version.

1. Fixed a problem where the name of a spectral grid for CMS-Wave was not preserved.
2. Fixed a problem where grid frames could “disappear” behind other data and not select correctly.
3. Fixed a crash that could happen when trying to convert a mesh to a map.
4. Changed SMS to allow small damping cellstrings for the BOUSS2D model.
5. Fixed a problem with generating cellstrings for BOUSS2D where no cellstring was created on the right side of the grid.
6. Fixed a problem with the generic model interface where boundary condition values were lost after opening and saving.
7. Fixed a crash that sometimes could happen when using texture mapping.
8. Fixed a crash that could happen when using the steering module with CMS-Flow/CMS-Wave.
9. Made it so you could replace the generic model definition being used without deleting the geometry.
10. Fixed a problem that caused element labels to print very small at times.
11. Fixed an issue where SMS would set the ADCIRC output files to binary if the output information wasn’t complete. This made it impossible to get back to ASCII files without editing the files by hand.

SMS 11.0.2

This is a bug-fix release for SMS 11.0. We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations. This version includes an update that can be installed on top of SMS 11.0.1 (sorry can't patch 11.0.0 due to an issue with our installation software).

Bugfixes

The following bugs have been fixed in this version.

1. Problem selecting cells in the spectral energy dialog.
2. Fixed a crash that could happen if you didn't have a scatter dataset and then converted it to mesh and then to map.
3. Fixed an issue where mesh elements were not displayed according to the setting in the display options dialog after generating a mesh using LTEA.
4. Fixed an issue where SMS stored the wrong directory if the user had to path to find the LeProvost files.
5. Fixed an issue with the LTEA mesh generation feature where SMS was not redistributing the ocean boundary based upon the user specification.
6. Fixed an issue where LTEA had spurious error messages pop-up during the meshing process.
7. SMS would sometimes write the wrong timestep when saving datasets to a tabular data file (*.txt).
8. AdH nodestring symbols were not displayed correctly after loading the simulation until the user went to the boundary condition dialog. This has been fixed.
9. SMS was not allowing CMS-Wave structure cells to have a negative modification value but this is valid in some cases.
10. Fixed an issue with the CMS-Wave model control parameters dealing with spreadsheet rows disappearing and not resizing correctly.
11. Fixed a crash that happened sometimes after duplicating a CMS-Wave grid.
12. Fixed a problem where the clock for first frame in a filmloop was not being displayed correctly if the starting time was not 0.0.
13. Fixed a problem where 1D *.mif files were not imported correctly into SMS when loading a *.tcf file.
14. Fixed a crash with running STWAVE full-plane after loading a project created with an earlier version of SMS.
15. SMS was not correctly interpreting some of the parameters correctly when importing STWAVE model files.
16. Fixed a hang when copy/pasting values into the time-series editor used with AdH.
17. STWAVE executable fixed to write datasets correctly.
18. Zonal classification could sometimes identify polygons incorrectly.
19. Auto-zmag was not updating when a new mesh was created until a frame or similar command was issued.
20. SMS was not correctly remembering specified raster projection after saving/loading a project.
21. In geographic projections, SMS was labeling x locations "east" that should have been labeled "west."
22. The raster contours sometimes changed after going to the display options dialog even when not changing the contour settings.
23. The 2D Gridframe was sometimes hidden behind raster.
24. Typo fixed in the menu for data calculator.
25. STWAVE boundary conditions dialog not "cutting" a row correctly.
26. Fixed crash when loading a project with 1D elements created in an earlier version of SMS.
27. Made it so you can specify paths to sediment datasets in *.xmdf file.
28. Fixed problem displaying functional surfaces with ati card.
29. Changed how LandXML files read to correctly identify "Northing" then "Easting."
30. SMS was incorrectly identifying elements as duplicates and removing them.
31. Fixed problem with local projection in *.mif/*.mid files.
32. Fixed crash when snapping two feature points.

33. STWAVE files not exported correctly when wind datasets being used.
34. Changed how we handle projections read when importing STWAVE model files so non-state plane coordinates are better handled.
35. Converting a raster to scatter created voids in the data.
36. Fixed saving tidal harmonics settings in ADCIRC model control.
37. You can now select all the points in a raster (before the top values sometimes weren't selecting).

SMS 11.0.1

This is a bug-fix release for SMS 11.0. We strongly recommend uninstalling any beta versions of SMS 11.0 before installing to avoid a potential problem with conflicting installations.

Bugfixes

The following bugs have been fixed in this version.

1. Fixed a bug where reprojecting a TUFLOW grid was not updating all of the z values correctly.
2. Fixed an issue where RMA2 mid-side nodes were not retaining their elevation data after saving/loading.
3. Fixed a couple minor issues with the Map->scatter dialog.
4. Made it so SMS would preserve the order of nodestrings from the AdH *.bc file.
5. Fixed a problem with ADCIRC fort.23 files that were reloading incorrectly.
6. Fixed a crash that could occur when copy/pasting when making irregular culverts in TUFLOW.
7. Fixed a problem where the functional surface legend was obscured by other data rather than being above the other data.

New Module – Raster Module

- You can now work with raster data (DEM) in SMS without needing to convert the data into a scatterset (TIN). Raster formats are more space efficient and quicker to draw than TIN based formats with the same number of points. This allows you to work with larger DEMs than previously possible within SMS.
- Rasters can be used to interpolate elevations to a scatter, mesh, or grid. You can create observation profile plots of raster data to see cross-section views of raster data.

New Interfaces

Planetary Boundary Layer (PBL)

- The PBL model is used to compute wind fields from tropical storms and hurricanes. The PBL engine is developed and maintained by Oceanweather Inc., experts in wind forecasting and hind casting. The PBL model takes an input storm track and outputs wind fields that can be used to force an ADCIRC model. Currently, distribution of the model itself is controlled by OWI. Negotiations to use the model, or get output files from the model, which can then be used in SMS as part of CSTORM or ADCIRC model runs, must go through OWI.

Adaptive Hydraulics (AdH)

- The 2D shallow water component of the ADaptive Hydraulics Modeling system (AdH) now has an interface within SMS. AdH was developed by the Coastal and Hydraulics Laboratory, ERDC, USACE (www.chl.erdc.usace.army.mil [1]). AdH solves the 2D shallow water equations, features an adaptive mesh solution to dynamically alter the resolution of the mesh based upon where it is needed, supports wetting and drying, boat effects, and wind effects. See the AdH website for more information on AdH (https://adh.usace.

army.mil/).

Note: The sediment and AD transport portions of AdH are not currently supported in the SMS interface.

WAM

- The global ocean WAve prediction Model called WAM is a third generation wave model. WAM predicts directional spectra as well as wave properties such as significant wave height, mean wave direction and frequency, swell wave height and mean direction, and wind stress fields corrected by including the wave induced stress and the drag coeffieient at each grid point at chosen output times. (http://chl.erdc.usace.army.mil/chl.aspx?p=s&a=software;8)
- The WAM interface in SMS supports building WAM grids, creating WAM simulations, nesting WAM grids, post-processing support (contours and spectra), and generating spectra for STWAVE input (requires the STWAVE interface).

Bouss2D Runup/Overtopping

- It is now possible to run Bouss2D in 1D mode to simulate run-up and overtopping. The runup/overtopping interface supports the ability to extract transects, position gages, specify roughness zones (Chezy or Manning), and define multiple wave cases. Post-processing includes 2D profile plots along transects, time-series plots of gage output, generated summary statistics such as height of highest 1/10, 1/50 of waves, and the point of furthest encroachment on each transect.

Other Additions and/or Changes

TUFLOW Advection/Diffusion Module

- There is a new Advection/Diffusion (AD) module for TUFLOW. TUFLOW AD simulates depth-averaged, two and one-dimensional constituent fate and transport. Both dissolved and particulate constituents can be simulated. TUFLOW AD adaptively expands its computational stencil (between third to ninth order) in areas where strong constituent gradients are identified. The module is fully supported within SMS including the ability to support spatially varied initial concentrations and transport coefficients.

Generic Model New Features

- We have added several features to the generic model interface to offer more options for providing a user-interface for models without a custom SMS interface.
- Now boundary condition and material parameters can now support more than just a floating point or curve value. Each parameter can be enumerated options, boolean, integer, float, curve, text, or float/curve (user chooses which).
- Now multiple boundary conditions can be assigned to the same entity (node, nodestring or elements). This is particularly handy if you have bc information for multiple solution types (flow and sediment transport).
- The generic model designer can show/hide parameters based upon an enumerated option parameter. For example, the user could choose between chezy and manning roughness approaches and show the appropriate parameters depending upon the choice. Boundary condition and Material properties can use global parameters or their own parameters to base the hide/show logic.
- The designer can also choose to have multiple material groups (roughness/sediment).
- In order to accomplish the features above and make parameters as consistent as possible, some existing card definitions have changed. This will necessity changes for models using the generic model interface and migrating

from SMS 10.1 to 1.0.0.

CMS Updates

- CMS Flow now supports the ability to run a coupled CMS Flow/CMS Wave model without the use of the steering module (inlined). Running the models inlined minimizes the file IO for the models decreasing runtimes.
- CMS Flow also supports an optional Implicit solution scheme allows for longer time-step sizes and parallel runs so you can distribute your work across all of your processor cores. The option to change which mode (2D or 3D) CMS Flow runs in has been removed. By default, CMS Flow will run in 2D.
- CMS Wave has several new features including the ability to define a muddy bed, non-linear wave effects, infragravity wave effects, spatially varied wind field, xmdf output, and a Gauss-seidel solution scheme that allows you to run across multiple processor cores.

TUFLOW Updates

- You can now import projects created outside of SMS. Multiple TCF files can be read together to maintain sharing of objects where supported by SMS. Makes it easy for someone familiar with SMS to pick up on a model started by someone else or to help transition to using TUFLOW within SMS.
- You can now choose to output datasets in XMDF format which is much faster (basically instant) for loading datasets into SMS. You can also add custom text to the output options to choose items not supported by SMS.
- You can now create, manage, and use irregular culverts from inside the SMS interface.
- You can now choose to have SMS write zpts to a new xf file format that is binary and very fast to read/write.
- Includes manhole support for the new TUFLOW manhole features. You can specify options for the automatically generated manholes and override these settings using a TUFLOW manhole coverage.
- You can now select grid cell locations (9 cell locations used by TUFLOW) using the “Feature objects->Select/Delete Data” command.

Cartesian grids store their own projection and reproject on the fly

- You can now have a projection associated with each of your cartesian grids. The grid will be reprojected on the fly into the working projection for display purposes. You can right-click on the grid and choose “Work in grid projection” to easily change your working projection to the grid’s projection which is required for working with the grid and using tools.

Note: Bouss2D and CMS-Wave do not currently have the ability to save this projection information.

STWAVE Updates

- We are now linked to STWAVE version 6 which has new file formats and improvements made by the model developers. You can now extract boundary conditions from a larger WAM run and have improved iteration control for the full-plane version. The new interface/model allows you to specify names for your boundary condition cases.

Improved Crash Reports

- The crash reports from SMS now contain more information often enough for our developers to fix the issue without any user interaction. Please allow SMS to post this information so our developers can fix issues that arise.

Dynamic background images from the web through ArcGIS

- If you have ArcGIS on your computer, you can use the GIS module within SMS to get background imagery that updates on the fly from the internet.

Particle module/PTM changes

- Particle sets can have a projection defined and are reprojected on the fly.
- You can estimate the number of particles that will be generated in by a PTM source coverage. This helps ensure you don’t accidentally generate so many particles that the computations take to long.
- Virtual gages - You can use virtual gages to determine concentrations and other data around a point or within a polygon.
- Particle filters - You can use particle filters to display a subset of particles to use for display, selection and compute grid datasets. You can use this to look at particles from particular sources, etc.

Bouss2D changes

- You can now have variable roughness defined by map polygons for a Bouss2D simulation

GenCade changes

- Wave gage event wave directions can now be specified in conventions other than shore normal, such as meteorological and oceanographic.
- Left and right bypass coefficients can be specified for inlets. The Y Left and Y Right have been removed and replaced by the left and right bypass coefficients.

General Features

- If you do file | Save as and choose an image file, SMS writes an associated world and projection file
- You can change the symbol size used in plots
- You can convert mesh elements to polygons using Mesh->Map. This makes it possible to get your mesh elements into a shapefile
- SMS now uses the existing background for flowtrace and drogue plots rather than have this specified separately
- “Zoom to” options to easily see specific information including: zoom to mesh, grid, scatter, and selections
- Option to always use white when printing rather than current background color
- You can now import TINs from LandXML files
- Find/select map points, arcs, and polygons by id (use zoom to selection to find in busy coverages)

References

[1] http://www.chl.erdc.usace.army.mil

What's New in SMS 10.1

We are excited about the new features in SMS 10.1. We have focused on updating portions of SMS which will benefit everyone. We have added new dataset creation options in the new Dataset Toolbox. We have added numerous new features for working with TINS in the scatter module including breaklines and several new import/conversion options. A full list of new features can be found below:

64-bit Version

SMS now distributes a 64-bit version. This version can access far more memory than the 32-bit version can so it is useful for working with large sets of data. The installation is the same for the 32 and 64 bit versions of SMS. You can choose the version to install if you have a 64-bit operating system. Both versions can be installed by running the installation twice. The performance (speed) of this version is basically the same as the 32-bit version. This version requires a 64-bit operating system (Vista-64 bit recommended).

General Features

- Dataset Toolbox – New dataset toolbox has new methods to create datasets as well as organizing some of the existing options. The new features include: creating comparison datasets, sampling datasets at specific times, computing derivatives or changes through time, and filtering datasets.
- Annotations – New annotation tools allow the creation of scale bars, North Arrows, screen space images for Logos, as well as rectangles, ovals, and lines in either screen or world space.
- Data Calculator – The data calculator now includes functions to take the average, minimum, or maximum of all the timesteps of a dataset and is included in the dataset toolbox.
- KMZ Filmloop Export – In addition to saving raster or vector data from SMS, you can now create animations that can be opened in Google Earth.
- Graphical Selection Tools – The modifier keys for graphical selection tools have been expanded and made consistent.
- Projections – SMS now includes many more GIS projection systems (previously referred to as Coordinate Systems) and can use projection (prj) files associated with images and GIS files. SMS can also create projection files.
- Image Projection Changes – Images now store their native projection and will automatically be displayed in the working projection.
- Measure Tool – SMS now includes a simple tool for measuring distances.
- Web Menu - New menu for accessing importing data from the web.
- Model Priority – There is a new preference that allows the user to specify the priority to launch numeric models.
- Vector Display Options – Vectors can now have their center or tip positions at the node or grid location being drawn at. This can be useful to prevent vector arrows to appear on land.
- Stick Plots – Vectors can now be displayed on specified points from a map coverage at points or along vertices on an arc. This can be useful for comparing data obtained along a transect.
- Save As Image – You can now save the contents of the graphics window to a jpeg or bitmap by doing a file save as and changing the save type.
- Remote desktop – In SMS 10.0, the screen would go blank if you started a remote session to look at an already running instance of SMS. This has been resolved.
- Edge Swap – You can now set a general preference to turn on/off automatic refreshing after an edge swap.

Module Specific Improvements

Cartesian Grid Module

- Interpolate Bathymetry to cells – You can now interpolate new bathymetry to the selected cells in a Cartesian grid without updating the entire grid.
- Smoothing – The smoothing options for a cartesian grid have been expanded to allow smoothing only on a portion of the grid.
- Duplicate Grid – You can duplicate an entire cartesian grid to create a second simulation that can be changed without altering the initial simulation.
- Cartesian Grid mapping – Map to grid attributes are now stored with the grid frames so it is easy to go back and create new grids with modified parameters (such as cell size).
- Transformed Cartesian Grid – You can transform an STWAVE or CMS-WAVE grid to realign it to the directions of the waves.
- Steering Module – The steering module now allows you to choose the grids to use when multiple cartesian grids exist.

Map Module

- Animating profile plots now supports multiple datasets – (before only the active dataset could be animated)
- Observation Profile Plot – Profile plots now have the ability to plot multiple time steps on a single plot.
- Feature Stamping – Feature stamps now automatically create breaklines appropriately when stamping to a scatter set (TIN).
- Spatial Data Coverage – New coverage type can be used to display compass plots at locations (such as wind velocities/directions).

Scatter Module (TINs)

- Scatter Breaklines – SMS now supports breaklines on scatter sets. Breaklines can be manually created, created from converting Map data (CAD or GIS to Map then scatter), created automatically when using feature stamping, or can be imported using the import wizard.
- Merge Scatter Sets – Merging scattersets can now preserve triangulation of initial scattersets where the scattersets to merge do not overlap.
- CAD/GIS faces to triangles – AutoCAD or ESRI 3D shapefiles with polygons representing faces can be converted directly to scatter triangles. This makes it easy to get AutoCAD or ESRI TIN data (export DWG/DXF or 3D Shapefile format) into SMS without having to retriangulate and fix the triangulation.
- Process Boundary Triangles – This new tool can be used to remove unwanted boundary triangles converted on the edge of the domain.
- Move Scatter Vertices – You can now unlock the scatter set using the vertex menu and move scatter vertices.
- File Import Wizard Additions for breaklines – Breaklines can be imported using the import wizard.

Individual Model Improvements

CMS

- CMS-Flow Salinity – CMS-Flow can now calculate salinity.
- CMS-Flow Advanced Cards – This dialog can be used to specify options that are supported within the CMS-Flow model but are not yet part of the SMS interface. Primarily useful for developers.
- CMS-Wave Simplified Formulation – New run mode for CMS-Wave that allows quicker runs for faster preliminary simulations.
- CMS-Wave Structure Types – New structure types with additional parameters that can be provided.

Generic Mesh Model

- 2D Mesh Files (*.2dm) – The generic model interface can now export full curves rather than sampling the curves based upon the timestep value.

STWAVE Model

- STWAVE Time Support – You can now reference input spectra to specific time values or have each spectra independent of time.

TABS Model

- RMA4 Element Loading – You can now add a mass loading directly to individual elements for an RMA4 advection/dispersion simulation.

TUFLOW

- TUFLOW ZShape – New coverage type used to modify geometry. It is similar to the geometry modification coverage but has more options including the ability to have the geometry vary with time.
- TUFLOW 2D Flow Constriction Shape coverage – New coverage type used to create 2D structures. These structure can be layered flow constrictions which allow flows both below and above a bridge deck.
- TUFLOW Inlet Database – For urban storm drain models, you can provide curves that describe the amount of flow captured by storm drain inlets based upon the depth of flow in the 2D cells rather than the shape options previously available.
- TUFLOW Network Node SX Additions – 1D nodes can be directly connected to a 2D domain without the need for SX lines or a connection coverage.
- TUFLOW Eddy Viscosity Options – The eddy viscosity used in the model can now be specified as a combination of a fixed eddy viscosity added to a computed smagorinsky viscosity value.

Features No Longer Supported

The models HECRAS and GENESIS are no longer supported in SMS. Our sister program WMS has an interface for HEC-RAS for those who are interested.

The RMA4 interface no longer supports BOD/DO since this functionality does not function correctly in RMA4.

What's New in SMS 10.0

What's New in SMS 10.0

General New Features:

- **Vista Support** – Text rendered in earlier versions of SMS did not display correctly when run under Windows Vista. This has been fixed in SMS 10.0.
- **Graphics Improvements** – The display pipeline has been completely overhauled in order to support hardware acceleration and reduce memory usage. In addition fill behind labels and aligning automatic contour labels with linear contours now work.
- **Improved DWG Support** – SMS 10.0 now supports AutoCAD files up to and including version 2008. In addition, AutoCAD files are displayed in 3D rather than 2D background data as in SMS 9.2.
- **KMZ File Export** – SMS can now export the currently displayed image as a raster with geo-referencing in a *.kmz file. Kmz files can be visualized inside of Google Earth [2].
- **Import Data From Web** – When importing data from Terraserver, SMS brings up a locator tool where you can locate the area to download using Microsoft Virtual Earth.
- **Arc Groups in Profile Plots** – When observation arcs are joined into arc groups, profile plots will join the data end to end rather than seeing separate curves. This allows the creation of a single curve in a profile plot from several arcs. Since each arc can have its own color it makes it easy to identify specific locations in the data.
- **Hide Unavailable Features and Unused Options** – The display options dialog by default only shows options for data that currently exist in SMS. When selecting a coverage type only coverages associated with registered features or models are put into the list. The preferences dialog also only works with registered features. These changes make it easier for users to find the options they want to use by hiding irrelevant features or options.
- **Filmloop Compression options** – You can now select a compression codec and associated quality in order to build smaller filmloop files.
- **Functional surface options** – It is now possible to color a functional surface using all of the options available for color filled contours.
- **Help system is a Wiki** – Rather than distribute chm files with SMS, the help is now found on a Wiki. This allows more people to get involved in updating the information. We are working hard to improve the help contents currently available.

Model Specific New Features:

ADCIRC

- **Spatial Attributes** – The ADCIRC interface now supports distributed spatial attributes (fort.13) files.

CGWAVE

- **Spatially varied bed friction and floating docks** – An updated version of CGWAVE (version 2.0) is now supported in SMS 10.0. This version includes options for spatially varied bed friction and floating docks.
- **Test Problems** - A new set of test problems is also provided to illustrate model capabilities

CMS-Flow (previously M2D)

- **Model Improvements** – The CMS-Flow interface in SMS 10.0 has been refreshed and updated to support for CMS-Flow v3.5. This version uses XMDF simulation files and can be run in explicit or implicit mode.
- **Interface Improvements** – Project management has been simplified, improved model parameter checking, and you can now work with input wave climate datasets outside the steering module.

PTM

- **Model Improvements –** The latest version of PTM (version 2.0) is now supported in SMS 10.0. PTM 2.0 supports hydrodynamic input from ADCIRC, ADCIRC3D, CH3D, CMS-Flow, AND CMS-Flow3D. Additional computation and output options are also available in PTM 2.0.
- **Interface Improvements –** The PTM Model Control has been redesigned. Mathematical operations can be performed on particle data sets using the Data Calculator. The particle display options have been expanded. New post processing features include creating data sets on a cartesian grid of particle count, accumulation, rate of accumulation, deposition, exposure, concentration, and dosage.

TUFLOW

- **Model Improvements –** A new boundary condition type has been added for a stage vs flow rating curve generated automatically from a water surface elevation slope.
- **Interface Improvements –** The TUFLOW interface for boundary conditions has been simplified for ease of use. The interface now supports the ability to generate and manage multiple 2D domains to allow for changes in resolution. The interface now also supports 2D flow constrictions to model bridges, peirs, or large culverts in 2D.

CMS-Wave (previously WABED)

- **Version/Feature Update –** SMS 10.0 interfaces with CMS-Wave v 1.67. This version of CMS-Wave includes functionality to allow wetting and drying, consider constant or spatially varied bed friction, and use constant or spatially varied forward and/or backward reflection. Parameters have also been added to allow user control of the intensity of diffraction and the type of wave breaking formula to use.

Models Removed From SMS

The following models are no longer available or supported in SMS 10.0:

- SED2D
- HIVEL

Article Sources and Contributors

What is SMS? *Source*: http://xmswiki.com/xms/index.php?oldid=41243 *Contributors*: Aanderson, Jcreer

Tutorials *Source*: http://xmswiki.com/xms/index.php?oldid=39802 *Contributors*: Jcreer, Jdhowlett

Sample Problems *Source*: http://xmswiki.com/xms/index.php?oldid=39800 *Contributors*: Jcreer, Jdhowlett

Highlights *Source*: http://xmswiki.com/xms/index.php?oldid=42578 *Contributors*: Jcreer, Rjones

SMS Screen *Source*: http://xmswiki.com/xms/index.php?oldid=41112 *Contributors*: Jcreer

What's New in SMS 11.1 *Source*: http://xmswiki.com/xms/index.php?oldid=42014 *Contributors*: Admin, Azundel, Jcreer, Tnewbold

Bugfixes SMS *Source*: http://xmswiki.com/xms/index.php?oldid=42902 *Contributors*: Jcreer, Jdhowlett, Rkelsch

Editing the view with the Mouse *Source*: http://xmswiki.com/xms/index.php?oldid=39810 *Contributors*: Abarney, Jcreer

FHWA:2010 Webinars *Source*: http://xmswiki.com/xms/index.php?oldid=39812 *Contributors*: Cbarlow, Jcreer

General Interface Features *Source*: http://xmswiki.com/xms/index.php?oldid=40967 *Contributors*: Jcreer, Rjones

Keyboard Shortcuts *Source*: http://xmswiki.com/xms/index.php?oldid=41647 *Contributors*: Jcreer, Jdhowlett

Publications *Source*: http://xmswiki.com/xms/index.php?oldid=39819 *Contributors*: Jcreer, Jdhowlett

Layout *Source*: http://xmswiki.com/xms/index.php?oldid=41612 *Contributors*: Jcreer, Jdhowlett

Data Toolbar *Source*: http://xmswiki.com/xms/index.php?oldid=40973 *Contributors*: Jcreer, Sramiah

Dynamic Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41614 *Contributors*: Jcreer, Jdhowlett

Edit Window *Source*: http://xmswiki.com/xms/index.php?oldid=40993 *Contributors*: Jcreer, Jdhowlett

Graphics Window *Source*: http://xmswiki.com/xms/index.php?oldid=41017 *Contributors*: Jcreer, Jdhowlett

Help or Status Window *Source*: http://xmswiki.com/xms/index.php?oldid=41013 *Contributors*: Jcreer, Jdhowlett

Macros *Source*: http://xmswiki.com/xms/index.php?oldid=42581 *Contributors*: Azundel, Jcreer, Jdhowlett

Menu Bar *Source*: http://xmswiki.com/xms/index.php?oldid=42612 *Contributors*: Azundel, Jcreer, Jdhowlett

Project Explorer *Source*: http://xmswiki.com/xms/index.php?oldid=42568 *Contributors*: Azundel, Jcreer

Project Explorer Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=42593 *Contributors*: Azundel, Jcreer, Sramiah

SMS Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41998 *Contributors*: Jcreer

Static Tools *Source*: http://xmswiki.com/xms/index.php?oldid=40956 *Contributors*: Jcreer, Jdhowlett

Time Step Window *Source*: http://xmswiki.com/xms/index.php?oldid=41113 *Contributors*: Jcreer, Jdhowlett

Toolbars *Source*: http://xmswiki.com/xms/index.php?oldid=40955 *Contributors*: Jcreer, Rjones

Support *Source*: http://xmswiki.com/xms/index.php?oldid=39834 *Contributors*: Jcreer

Downloads *Source*: http://xmswiki.com/xms/index.php?oldid=39835 *Contributors*: Jcreer, Rjones

FTP Site Info *Source*: http://xmswiki.com/xms/index.php?oldid=38023 *Contributors*: Abarney, Jcreer

License Agreement *Source*: http://xmswiki.com/xms/index.php?oldid=39837 *Contributors*: Jcreer, Ksmith

Registering SMS *Source*: http://xmswiki.com/xms/index.php?oldid=43452 *Contributors*: Jcreer

Hardware Locks *Source*: http://xmswiki.com/xms/index.php?oldid=43474 *Contributors*: Jcreer

Graphics Card Troubleshooting *Source*: http://xmswiki.com/xms/index.php?oldid=39838 *Contributors*: Jcreer, Ksmith

Setting up SMS – Introduction *Source*: http://xmswiki.com/xms/index.php?oldid=43459 *Contributors*: Jcreer, Jdhowlett

System Requirements *Source*: http://xmswiki.com/xms/index.php?oldid=28735 *Contributors*: Jcreer, Ksmith, Morelandtom, Wood

Breaklines *Source*: http://xmswiki.com/xms/index.php?oldid=41649 *Contributors*: Azundel, Jcreer

Object Info *Source*: http://xmswiki.com/xms/index.php?oldid=42095 *Contributors*: Jcreer

Select/Delete Data... *Source*: http://xmswiki.com/xms/index.php?oldid=41554 *Contributors*: Jcreer

Plot Window *Source*: http://xmswiki.com/xms/index.php?oldid=41988 *Contributors*: Azundel, Jcreer, Rjones

Computed vs. Observed Data *Source*: http://xmswiki.com/xms/index.php?oldid=41977 *Contributors*: Jcreer, Jdhowlett

Error Summary *Source*: http://xmswiki.com/xms/index.php?oldid=41992 *Contributors*: Jcreer, Jdhowlett

Error vs. Simulation Plot *Source*: http://xmswiki.com/xms/index.php?oldid=41980 *Contributors*: Jcreer, Jdhowlett

Error vs. Time Step Plot *Source*: http://xmswiki.com/xms/index.php?oldid=41981 *Contributors*: Jcreer, Jdhowlett

Observation Profile *Source*: http://xmswiki.com/xms/index.php?oldid=41990 *Contributors*: Jcreer, Jdhowlett

Profile Customization Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41989 *Contributors*: Jcreer, Jdhowlett

Residual vs. Observed Data *Source*: http://xmswiki.com/xms/index.php?oldid=41979 *Contributors*: Jcreer, Jdhowlett

Time Series *Source*: http://xmswiki.com/xms/index.php?oldid=39854 *Contributors*: Jcreer, Jdhowlett

Time Series Data File *Source*: http://xmswiki.com/xms/index.php?oldid=39855 *Contributors*: Jcreer, Jdhowlett

Time Series Plot *Source*: http://xmswiki.com/xms/index.php?oldid=41993 *Contributors*: Jcreer, Jdhowlett

View Values Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41983 *Contributors*: Jcreer, Jdhowlett

ARR Mesh Quality Assessment Plot *Source*: http://xmswiki.com/xms/index.php?oldid=42582 *Contributors*: Jcreer

Animations *Source*: http://xmswiki.com/xms/index.php?oldid=41673 *Contributors*: Jcreer, Rjones

Film Loop Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=41658 *Contributors*: Bmerrell, Jcreer

Film Loop Drogue Plot Options *Source*: http://xmswiki.com/xms/index.php?oldid=41660 *Contributors*: Bmerrell, Jcreer

Film Loop Flow Trace Options *Source*: http://xmswiki.com/xms/index.php?oldid=41668 *Contributors*: Bmerrell, Jcreer

Film Loop General Options *Source*: http://xmswiki.com/xms/index.php?oldid=41663 *Contributors*: Jcreer, Jdhowlett

Film Loop Multiple Views *Source*: http://xmswiki.com/xms/index.php?oldid=41670 *Contributors*: Bmerrell, Jcreer

Film Loop Time Step Options *Source*: http://xmswiki.com/xms/index.php?oldid=41665 *Contributors*: Jcreer, Jdhowlett

CAD Data *Source*: http://xmswiki.com/xms/index.php?oldid=41228 *Contributors*: Azundel, Jcreer

CAD Data Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41230 *Contributors*: Jcreer, Jdhowlett

Coordinate Systems *Source*: http://xmswiki.com/xms/index.php?oldid=41675 *Contributors*: Jcreer, Ksmith

Coordinate Conversions *Source*: http://xmswiki.com/xms/index.php?oldid=37513 *Contributors*: Ksmith

CPP Coordinate System *Source*: http://xmswiki.com/xms/index.php?oldid=37717 *Contributors*: Ksmith

Geographic Coordinate System *Source*: http://xmswiki.com/xms/index.php?oldid=36074 *Contributors*: Ksmith, Unjedai

Local Coordinate System *Source*: http://xmswiki.com/xms/index.php?oldid=38290 *Contributors*: Jcreer, Ksmith

Single Point Conversion *Source*: http://xmswiki.com/xms/index.php?oldid=41732 *Contributors*: Jcreer, Unjedai

Projections *Source*: http://xmswiki.com/xms/index.php?oldid=37514 *Contributors*: Ksmith, Unjedai

UTM Africa *Source*: http://xmswiki.com/xms/index.php?oldid=28045 *Contributors*: Jdhowlett

UTM Asia *Source*: http://xmswiki.com/xms/index.php?oldid=28049 *Contributors*: Jdhowlett

UTM Australia *Source*: http://xmswiki.com/xms/index.php?oldid=28050 *Contributors*: Jdhowlett

UTM Coordinate System *Source*: http://xmswiki.com/xms/index.php?oldid=30320 *Contributors*: Jdhowlett

UTM Europe *Source*: http://xmswiki.com/xms/index.php?oldid=28051 *Contributors*: Jdhowlett

UTM North America *Source*: http://xmswiki.com/xms/index.php?oldid=28052 *Contributors*: Jdhowlett

UTM South American *Source*: http://xmswiki.com/xms/index.php?oldid=28053 *Contributors*: Jdhowlett

State Plane Coordinate System *Source*: http://xmswiki.com/xms/index.php?oldid=28041 *Contributors*: Jdhowlett

Alaska State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28054 *Contributors*: Jdhowlett

Hawaii State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28055 *Contributors*: Jdhowlett

Mideast State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28056 *Contributors*: Jdhowlett

Midwest State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28059 *Contributors*: Jdhowlett

New England State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28057 *Contributors*: Jdhowlett

Northwest State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28060 *Contributors*: Jdhowlett

South Central State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28061 *Contributors*: Jdhowlett

South East State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28062 *Contributors*: Jdhowlett

Southwest State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=28063 *Contributors*: Jdhowlett

Virginia Area State Plane *Source*: http://xmswiki.com/xms/index.php?oldid=38291 *Contributors*: Jcreer

Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=42596 *Contributors*: Azundel, Jcreer, Jdhowlett

Create Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=41847 *Contributors*: Azundel, Jcreer

Dataset Toolbox *Source*: http://xmswiki.com/xms/index.php?oldid=41685 *Contributors*: Jcreer, Jdhowlett

Data Calculator *Source*: http://xmswiki.com/xms/index.php?oldid=43080 *Contributors*: Jcreer, Jdhowlett, Ksmith

Size Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=42759 *Contributors*: Jcreer, Jdhowlett

Smooth Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=41687 *Contributors*: Jcreer, Twood, Wood

Metadata *Source*: http://xmswiki.com/xms/index.php?oldid=42616 *Contributors*: Azundel, Jcreer

Dataset Info *Source*: http://xmswiki.com/xms/index.php?oldid=42614 *Contributors*: Azundel, Jcreer

Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42560 *Contributors*: Jcreer, Sramiah

3D Visualization *Source*: http://xmswiki.com/xms/index.php?oldid=42559 *Contributors*: Jcreer, Rjones

Color Options *Source*: http://xmswiki.com/xms/index.php?oldid=42574 *Contributors*: Jcreer, Jdhowlett

Functional Surfaces *Source*: http://xmswiki.com/xms/index.php?oldid=41744 *Contributors*: Jcreer, Jdhowlett

Lighting Options *Source*: http://xmswiki.com/xms/index.php?oldid=39891 *Contributors*: Cdickson, Jcreer, Ksmith

Raster Options *Source*: http://xmswiki.com/xms/index.php?oldid=39892 *Contributors*: Cdickson, Jcreer, Wood

General Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=39893 *Contributors*: Jcreer, Jdhowlett

Z Magnification *Source*: http://xmswiki.com/xms/index.php?oldid=39894 *Contributors*: Jcreer

Contour Options *Source*: http://xmswiki.com/xms/index.php?oldid=42554 *Contributors*: Jcreer, Jdhowlett

New Palette *Source*: http://xmswiki.com/xms/index.php?oldid=42563 *Contributors*: Jcreer, Jdhowlett

Vector Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42561 *Contributors*: Jcreer, Jdhowlett

Visualization for 3D Solutions *Source*: http://xmswiki.com/xms/index.php?oldid=42566 *Contributors*: Jcreer

Export Tabular File *Source*: http://xmswiki.com/xms/index.php?oldid=39904 *Contributors*: Azundel, Jcreer

Exporting Profile Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=42585 *Contributors*: Jcreer, Jdhowlett

Export Meshes in MIF/MID Format *Source*: http://xmswiki.com/xms/index.php?oldid=41747 *Contributors*: Jcreer

Export Dataset Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41758 *Contributors*: Jcreer, Jdhowlett

Export Map Data in Shape Format *Source*: http://xmswiki.com/xms/index.php?oldid=39909 *Contributors*: Jcreer

Data transform *Source*: http://xmswiki.com/xms/index.php?oldid=41843 *Contributors*: Jcreer, Jdhowlett

Zonal Classification *Source*: http://xmswiki.com/xms/index.php?oldid=41800 *Contributors*: Jcreer, Rjones

Images *Source*: http://xmswiki.com/xms/index.php?oldid=41736 *Contributors*: Jcreer, Wood

Image Pyramids *Source*: http://xmswiki.com/xms/index.php?oldid=41737 *Contributors*: Jcreer, Jdhowlett

Import from Web *Source*: http://xmswiki.com/xms/index.php?oldid=43434 *Contributors*: Cmsmemoe, Jcreer, Ksmith, Unjedai, Wood

Registering an Image *Source*: http://xmswiki.com/xms/index.php?oldid=39779 *Contributors*: Jcreer, Ksmith

Save as Image *Source*: http://xmswiki.com/xms/index.php?oldid=39940 *Contributors*: Jcreer, Sramiah

Web Service for Background Imagery *Source*: http://xmswiki.com/xms/index.php?oldid=39941 *Contributors*: Jcreer

Preferences *Source*: http://xmswiki.com/xms/index.php?oldid=41741 *Contributors*: Jcreer, Rjones

Time Settings *Source*: http://xmswiki.com/xms/index.php?oldid=41597 *Contributors*: Jcreer, Jdhowlett

Spectral Energy *Source*: http://xmswiki.com/xms/index.php?oldid=39944 *Contributors*: Jcreer, Jdhowlett

Create Spectral Energy Grid *Source*: http://xmswiki.com/xms/index.php?oldid=39945 *Contributors*: Abarney, Jcreer

Generate/Edit Spectra *Source*: http://xmswiki.com/xms/index.php?oldid=39946 *Contributors*: Jcreer, Jschultze

Import Spectra *Source*: http://xmswiki.com/xms/index.php?oldid=39947 *Contributors*: Jcreer, Jdhowlett

Spectral Grid Properties *Source*: http://xmswiki.com/xms/index.php?oldid=39948 *Contributors*: Abarney, Jcreer

Datasets VTK *Source*: http://xmswiki.com/xms/index.php?oldid=41707 *Contributors*: Jcreer

Conversions Scalar↔Vector *Source*: http://xmswiki.com/xms/index.php?oldid=41232 *Contributors*: Jcreer

Dataset Calculator VTK *Source*: http://xmswiki.com/xms/index.php?oldid=41700 *Contributors*: Jcreer

Interpolation VTK *Source*: http://xmswiki.com/xms/index.php?oldid=41704 *Contributors*: Jcreer

Modules *Source*: http://xmswiki.com/xms/index.php?oldid=40966 *Contributors*: Azundel, Jcreer

Annotations *Source*: http://xmswiki.com/xms/index.php?oldid=41245 *Contributors*: Jcreer, Ksmith

1D Grid Module *Source*: http://xmswiki.com/xms/index.php?oldid=41759 *Contributors*: Jcreer, Jdhowlett

1D Grid Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42096 *Contributors*: Jcreer, UserRightsTester

1D Grid Tools *Source*: http://xmswiki.com/xms/index.php?oldid=42526 *Contributors*: Azundel, Jcreer, Jdhowlett

1D River Module *Source*: http://xmswiki.com/xms/index.php?oldid=42109 *Contributors*: Jcreer, Jdhowlett

1D River Hydraulics Data Browser *Source*: http://xmswiki.com/xms/index.php?oldid=41335 *Contributors*: Abarney, Jcreer

1D River Hydraulics Profile Plots *Source*: http://xmswiki.com/xms/index.php?oldid=41994 *Contributors*: Abarney, Jcreer

1D River Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41340 *Contributors*: Jcreer, Jdhowlett

1D River Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41853 *Contributors*: Jcreer, Jdhowlett

Delete Simulation *Source*: http://xmswiki.com/xms/index.php?oldid=41343 *Contributors*: Abarney, Jcreer

Editing Cross Sections *Source*: http://xmswiki.com/xms/index.php?oldid=41453 *Contributors*: Jcreer, Jdhowlett

Extracting Cross Sections *Source*: http://xmswiki.com/xms/index.php?oldid=41344 *Contributors*: Jcreer, Jdhowlett

Interpolate Cross Sections *Source*: http://xmswiki.com/xms/index.php?oldid=41345 *Contributors*: Jcreer, Jdhowlett

Managing Cross Sections *Source*: http://xmswiki.com/xms/index.php?oldid=41346 *Contributors*: Jcreer, Jdhowlett

Recompute All Stations *Source*: http://xmswiki.com/xms/index.php?oldid=41347 *Contributors*: Jcreer, Jdhowlett

RivHyd Data Export *Source*: http://xmswiki.com/xms/index.php?oldid=41348 *Contributors*: Cballard, Jcreer

RivHyd Post-Processing *Source*: http://xmswiki.com/xms/index.php?oldid=42002 *Contributors*: Abarney, Jcreer

3D Cartesian Grid Module *Source*: http://xmswiki.com/xms/index.php?oldid=39966 *Contributors*: Jcreer, Jdhowlett

3D Cartesian Grid Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=41598 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Module *Source*: http://xmswiki.com/xms/index.php?oldid=41764 *Contributors*: Azundel, Jcreer

Cartesian Grid Module Overview *Source*: http://xmswiki.com/xms/index.php?oldid=42564 *Contributors*: Jcreer

2D Grid Files *Source*: http://xmswiki.com/xms/index.php?oldid=14618 *Contributors*: Jcreer, Mwtoews

Cartesian Grid Coordinates *Source*: http://xmswiki.com/xms/index.php?oldid=41322 *Contributors*: Jcreer

Cartesian Grid Data Menu *Source*: http://xmswiki.com/xms/index.php?oldid=42577 *Contributors*: Azundel, Jcreer, Jdhowlett

Cartesian Grid to Map *Source*: http://xmswiki.com/xms/index.php?oldid=42503 *Contributors*: Azundel

Cartesian Grid to Scatter *Source*: http://xmswiki.com/xms/index.php?oldid=42501 *Contributors*: Azundel, Jdhowlett

SMS:Cartesian Grid to Mesh *Source*: http://xmswiki.com/xms/index.php?oldid=42500 *Contributors*: Azundel

Cartesian Grid Find Cell *Source*: http://xmswiki.com/xms/index.php?oldid=41323 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=41324 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Module Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41766 *Contributors*: Jcreer, Jdhowlett

Cartesian Grid Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41326 *Contributors*: Jcreer, Jdhowlett

Grid Frame *Source*: http://xmswiki.com/xms/index.php?oldid=41499 *Contributors*: Abarney, Jcreer

Grid Smoothing *Source*: http://xmswiki.com/xms/index.php?oldid=41328 *Contributors*: Jcreer, Jdhowlett

Refine Point Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41935 *Contributors*: Jcreer, Jdhowlett

Curvilinear Grid Module *Source*: http://xmswiki.com/xms/index.php?oldid=41779 *Contributors*: Jcreer

Curvilinear Grid Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42565 *Contributors*: Jcreer

Curvilinear Grid Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41768 *Contributors*: Jcreer

Curvilinear Grid Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41770 *Contributors*: Jcreer

GIS Module *Source*: http://xmswiki.com/xms/index.php?oldid=41767 *Contributors*: Jcreer, Rjones

Importing Shapefiles *Source*: http://xmswiki.com/xms/index.php?oldid=41308 *Contributors*: Abarney, Jcreer

GIS Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41310 *Contributors*: Jcreer, Jdhowlett

GIS Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41306 *Contributors*: Jcreer, Jdhowlett

GIS Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42570 *Contributors*: Jcreer, Jdhowlett

ArcObjects *Source*: http://xmswiki.com/xms/index.php?oldid=41317 *Contributors*: Jcreer, Jdhowlett

GIS to Feature Objects Wizard *Source*: http://xmswiki.com/xms/index.php?oldid=41316 *Contributors*: Jcreer, Jdhowlett

GIS Module Right Click Commands *Source*: http://xmswiki.com/xms/index.php?oldid=41313 *Contributors*: Jcreer

Map Module *Source*: http://xmswiki.com/xms/index.php?oldid=41780 *Contributors*: Azundel, Jcreer

Map Module Overview *Source*: http://xmswiki.com/xms/index.php?oldid=40284 *Contributors*: Jcreer

Grid Frame *Source*: http://xmswiki.com/xms/index.php?oldid=41499 *Contributors*: Abarney, Jcreer

Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=41782 *Contributors*: Azundel, Jcreer

Generic Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=39992 *Contributors*: Jcreer

Area Property Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=41784 *Contributors*: Jcreer, Jdhowlett

Feature Stamping *Source*: http://xmswiki.com/xms/index.php?oldid=39988 *Contributors*: Jcreer, Jdhowlett

Observations *Source*: http://xmswiki.com/xms/index.php?oldid=41996 *Contributors*: Jcreer, Jschultze

Particle/Drogue *Source*: http://xmswiki.com/xms/index.php?oldid=39989 *Contributors*: Azundel, Jcreer

Spatial Data *Source*: http://xmswiki.com/xms/index.php?oldid=41529 *Contributors*: Jcreer, Jdhowlett

Shoals *Source*: http://xmswiki.com/xms/index.php?oldid=41895 *Contributors*: Azundel, Jcreer

Model Specific Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=41067 *Contributors*: Jcreer

1D Hyd Cross-Section Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40003 *Contributors*: Jcreer, Jdhowlett

1D River Conceptual Model *Source*: http://xmswiki.com/xms/index.php?oldid=41929 *Contributors*: Jcreer, Jdhowlett

ADCIRC *Source*: http://xmswiki.com/xms/index.php?oldid=40005 *Contributors*: Jcreer, Jdhowlett

ADCIRC Wind Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40019 *Contributors*: Jcreer

ADH *Source*: http://xmswiki.com/xms/index.php?oldid=40006 *Contributors*: Jcreer, Sramiah

Dredging *Source*: http://xmswiki.com/xms/index.php?oldid=40741 *Contributors*: Jcreer, Sramiah

EFDC *Source*: http://xmswiki.com/xms/index.php?oldid=41496 *Contributors*: Jcreer

ESMF – Earth System Modeling Framework *Source*: http://xmswiki.com/xms/index.php?oldid=41497 *Contributors*: Jcreer

FATE *Source*: http://xmswiki.com/xms/index.php?oldid=40742 *Contributors*: Jcreer, Sramiah

GenCade *Source*: http://xmswiki.com/xms/index.php?oldid=40013 *Contributors*: Jcreer

Generic Mesh Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40015 *Contributors*: Jcreer

LTFate *Source*: http://xmswiki.com/xms/index.php?oldid=40743 *Contributors*: Jcreer, Sramiah

Synthetic Storm Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40021 *Contributors*: Jcreer

TUFLOW Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=41468 *Contributors*: Jcreer, Jdhowlett

Interface Components *Source*: http://xmswiki.com/xms/index.php?oldid=40053 *Contributors*: Jcreer

Map Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=40984 *Contributors*: Jcreer, Jdhowlett

Map Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=40183 *Contributors*: Jcreer, Jdhowlett

Map Feature Objects Menu *Source*: http://xmswiki.com/xms/index.php?oldid=43154 *Contributors*: Jcreer, Rjones

Map Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41902 *Contributors*: Jcreer, Jdhowlett

Project Explorer Items *Source*: http://xmswiki.com/xms/index.php?oldid=41901 *Contributors*: Jcreer

Feature Objects Types *Source*: http://xmswiki.com/xms/index.php?oldid=38901 *Contributors*: Jcreer, Sramiah

Attributes in Feature Objects Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41957 *Contributors*: Jcreer, Jdhowlett

Find (By ID) *Source*: http://xmswiki.com/xms/index.php?oldid=40063 *Contributors*: Jcreer

Export (Convert to CAD) *Source*: http://xmswiki.com/xms/index.php?oldid=40070 *Contributors*: Jcreer

Selection *Source*: http://xmswiki.com/xms/index.php?oldid=41166 *Contributors*: Jcreer

Digitize *Source*: http://xmswiki.com/xms/index.php?oldid=40064 *Contributors*: Jcreer

Import Options *Source*: http://xmswiki.com/xms/index.php?oldid=40065 *Contributors*: Jcreer

Import From CAD/GIS *Source*: http://xmswiki.com/xms/index.php?oldid=40068 *Contributors*: Jcreer

Convert From Mesh/Scatter *Source*: http://xmswiki.com/xms/index.php?oldid=40067 *Contributors*: Jcreer

Build Polygons *Source*: http://xmswiki.com/xms/index.php?oldid=41466 *Contributors*: Jcreer

All *Source*: http://xmswiki.com/xms/index.php?oldid=41546 *Contributors*: Jcreer

Converting Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=40072 *Contributors*: Jcreer

Unstructured Grid Generation from a Conceptual Model *Source*: http://xmswiki.com/xms/index.php?oldid=40074 *Contributors*: Jcreer

Nodes/Vertices *Source*: http://xmswiki.com/xms/index.php?oldid=40073 *Contributors*: Jcreer

Select/Delete Data... *Source*: http://xmswiki.com/xms/index.php?oldid=41554 *Contributors*: Jcreer

Arcs *Source*: http://xmswiki.com/xms/index.php?oldid=41823 *Contributors*: Azundel, Jcreer

Feature Object Commands *Source*: http://xmswiki.com/xms/index.php?oldid=41483 *Contributors*: Jcreer

Mesh Module *Source*: http://xmswiki.com/xms/index.php?oldid=40089 *Contributors*: Jcreer, Jschultze

Mesh Generation *Source*: http://xmswiki.com/xms/index.php?oldid=40090 *Contributors*: Jcreer, Jdhowlett

Refine Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40823 *Contributors*: Jcreer, Jdhowlett

Refine Point Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41935 *Contributors*: Jcreer, Jdhowlett

2D Mesh Options Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40093 *Contributors*: Jcreer, Jdhowlett

2D Mesh Polygon Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40688 *Contributors*: Jcreer, Jdhowlett

Advancing Front Triangulation *Source*: http://xmswiki.com/xms/index.php?oldid=40095 *Contributors*: Jcreer, Jdhowlett

Mesh Node Triangulation *Source*: http://xmswiki.com/xms/index.php?oldid=40096 *Contributors*: Jcreer, Jdhowlett

Patch *Source*: http://xmswiki.com/xms/index.php?oldid=40097 *Contributors*: Jcreer, Jdhowlett

Patches *Source*: http://xmswiki.com/xms/index.php?oldid=40098 *Contributors*: Jcreer, Jdhowlett

Paving *Source*: http://xmswiki.com/xms/index.php?oldid=40099 *Contributors*: Jcreer, Jdhowlett

Rectangular Patch *Source*: http://xmswiki.com/xms/index.php?oldid=40100 *Contributors*: Jcreer, Jdhowlett

Scalar Paving Density *Source*: http://xmswiki.com/xms/index.php?oldid=26865 *Contributors*: Jcreer, Jdhowlett

Triangular Patch *Source*: http://xmswiki.com/xms/index.php?oldid=40103 *Contributors*: Jcreer, Jdhowlett

Adaptive Tesselation *Source*: http://xmswiki.com/xms/index.php?oldid=42031 *Contributors*: Jcreer, Jdhowlett

Mesh Project Explorer Items *Source*: http://xmswiki.com/xms/index.php?oldid=41907 *Contributors*: Jcreer

Mesh Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=42572 *Contributors*: Bmerrell, Jcreer

Mesh Quality *Source*: http://xmswiki.com/xms/index.php?oldid=40105 *Contributors*: Jcreer, Jdhowlett

2D Mesh Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=40959 *Contributors*: Jcreer, Jdhowlett

Editing 2D Meshes *Source*: http://xmswiki.com/xms/index.php?oldid=41962 *Contributors*: Jcreer, Jdhowlett

Mesh Nodestring Boundary Conditions Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40106 *Contributors*: Jcreer, Jdhowlett

2D Mesh Module Tools Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41856 *Contributors*: Jcreer

2D Mesh Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41190 *Contributors*: Jcreer, Jdhowlett

2D Mesh Nodestrings Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41920 *Contributors*: Jcreer, Jdhowlett

Mesh Data Menu *Source*: http://xmswiki.com/xms/index.php?oldid=42556 *Contributors*: Jcreer, Jdhowlett

2D Mesh Elements Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41715 *Contributors*: Jcreer, Jdhowlett

Element types *Source*: http://xmswiki.com/xms/index.php?oldid=39813 *Contributors*: Jcreer, Unjedai

Assign Material Type *Source*: http://xmswiki.com/xms/index.php?oldid=41960 *Contributors*: Jcreer, Jdhowlett

Boundary Triangles *Source*: http://xmswiki.com/xms/index.php?oldid=41961 *Contributors*: Abarney, Jcreer

Convert Elements *Source*: http://xmswiki.com/xms/index.php?oldid=41711 *Contributors*: Jcreer, Jdhowlett

Find Element *Source*: http://xmswiki.com/xms/index.php?oldid=41714 *Contributors*: Jcreer, Jdhowlett

Mesh Element Options *Source*: http://xmswiki.com/xms/index.php?oldid=41963 *Contributors*: Jcreer, Jdhowlett

Relax Elements *Source*: http://xmswiki.com/xms/index.php?oldid=41688 *Contributors*: Jcreer, Jdhowlett

Fix Bad Area Transitions *Source*: http://xmswiki.com/xms/index.php?oldid=40116 *Contributors*: Jcreer

2D Mesh Nodes Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41812 *Contributors*: Azundel, Jcreer, Jdhowlett

2D Mesh Node Options Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41958 *Contributors*: Jcreer, Jdhowlett

Find 2D Mesh Node Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40117 *Contributors*: Jcreer, Jdhowlett

Renumber *Source*: http://xmswiki.com/xms/index.php?oldid=41959 *Contributors*: Jcreer, Rjones

Reduce Nodal Connectivity *Source*: http://xmswiki.com/xms/index.php?oldid=41819 *Contributors*: Azundel, Jcreer

Particle Module *Source*: http://xmswiki.com/xms/index.php?oldid=40130 *Contributors*: Jcreer, Rjones

Particle Data Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40126 *Contributors*: Jcreer, Jdhowlett

Particle Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=40986 *Contributors*: Jcreer, Jdhowlett

Particle Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=40168 *Contributors*: Jcreer, Jdhowlett

Particle Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=40965 *Contributors*: Jcreer, Jdhowlett

Particle Report *Source*: http://xmswiki.com/xms/index.php?oldid=40131 *Contributors*: Jcreer, Rjones

Extract Particle Subset *Source*: http://xmswiki.com/xms/index.php?oldid=40132 *Contributors*: Jcreer

Particle Grid Dataset Bin Elevations *Source*: http://xmswiki.com/xms/index.php?oldid=40133 *Contributors*: Jcreer

Particle Module Compute Grid Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=41073 *Contributors*: Bmerrell, Jcreer

Particle Module Create Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=42066 *Contributors*: Jcreer, Jdhowlett

PTM Create Grid Datasets – Fence Diagrams *Source*: http://xmswiki.com/xms/index.php?oldid=40135 *Contributors*: Jcreer

Scatter Module *Source*: http://xmswiki.com/xms/index.php?oldid=41608 *Contributors*: Jcreer, Wood

Scatter Data Tools *Source*: http://xmswiki.com/xms/index.php?oldid=40137 *Contributors*: Cballard, Jcreer

Scatter Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41145 *Contributors*: Jcreer, Jdhowlett

Scatter Triangles Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41729 *Contributors*: Jcreer, Jdhowlett

Scatter Vertices Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40140 *Contributors*: Jcreer, Jdhowlett

Transform *Source*: http://xmswiki.com/xms/index.php?oldid=41716 *Contributors*: Jcreer

Interface Components *Source*: http://xmswiki.com/xms/index.php?oldid=39046 *Contributors*: Jcreer

Scatter Module Menus *Source*: http://xmswiki.com/xms/index.php?oldid=40185 *Contributors*: Jcreer, Jdhowlett

Scatter Module Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=40983 *Contributors*: Jcreer, Jdhowlett

Scatter Module Tools *Source*: http://xmswiki.com/xms/index.php?oldid=40961 *Contributors*: Jcreer, Jdhowlett

Scatter Module Tools Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41146 *Contributors*: Jcreer

Project Explorer Items *Source*: http://xmswiki.com/xms/index.php?oldid=39052 *Contributors*: Jcreer

Scatter Module Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41900 *Contributors*: Jcreer, Jdhowlett

Scalar Value Options *Source*: http://xmswiki.com/xms/index.php?oldid=42032 *Contributors*: Jcreer, Jdhowlett

Data Module Commands *Source*: http://xmswiki.com/xms/index.php?oldid=41196 *Contributors*: Jcreer, Unjedai

Scatter Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=39090 *Contributors*: Jcreer

Grid Frame Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41246 *Contributors*: Jcreer, Jdhowlett

Use of DEMs in the Scatter Module *Source*: http://xmswiki.com/xms/index.php?oldid=41605 *Contributors*: Cballard, Jcreer

Scatter Options *Source*: http://xmswiki.com/xms/index.php?oldid=41125 *Contributors*: Jcreer, Jschultze

Scatter Breakline Options *Source*: http://xmswiki.com/xms/index.php?oldid=42055 *Contributors*: Azundel, Jcreer

Scatter Breakline Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41122 *Contributors*: Jcreer, Jdhowlett

Process Boundary Triangles *Source*: http://xmswiki.com/xms/index.php?oldid=42094 *Contributors*: Jcreer, Jdhowlett

Interpolate to Scatter Set *Source*: http://xmswiki.com/xms/index.php?oldid=41132 *Contributors*: Jcreer, Jdhowlett

Generate Contour Breaklines *Source*: http://xmswiki.com/xms/index.php?oldid=39097 *Contributors*: Jcreer

Scatter Data Menu *Source*: http://xmswiki.com/xms/index.php?oldid=42558 *Contributors*: Jcreer, Jdhowlett

Scatter Filter *Source*: http://xmswiki.com/xms/index.php?oldid=39104 *Contributors*: Jcreer

Scatter Interpolation *Source*: http://xmswiki.com/xms/index.php?oldid=40141 *Contributors*: Jcreer, Jdhowlett

Laplacian Interpolation *Source*: http://xmswiki.com/xms/index.php?oldid=40142 *Contributors*: Jcreer

Inverse Distance Weighted Interpolation *Source*: http://xmswiki.com/xms/index.php?oldid=40143 *Contributors*: Abarney, Jcreer

Natural Neighbor Interpolation *Source*: http://xmswiki.com/xms/index.php?oldid=40144 *Contributors*: Jcreer, Jdhowlett

Linear Interpolation *Source*: http://xmswiki.com/xms/index.php?oldid=41087 *Contributors*: Abarney, Jcreer

Historical Surveys *Source*: http://xmswiki.com/xms/index.php?oldid=41694 *Contributors*: Abarney, Jcreer

Electronic Charts *Source*: http://xmswiki.com/xms/index.php?oldid=41697 *Contributors*: Abarney, Jcreer

Raster Module *Source*: http://xmswiki.com/xms/index.php?oldid=41384 *Contributors*: Jcreer

Raster Functionalities *Source*: http://xmswiki.com/xms/index.php?oldid=42584 *Contributors*: Jcreer

Raster Module Interface *Source*: http://xmswiki.com/xms/index.php?oldid=41387 *Contributors*: Jcreer

Raster Right Click Menus *Source*: http://xmswiki.com/xms/index.php?oldid=41391 *Contributors*: Jcreer

Raster Menu Items *Source*: http://xmswiki.com/xms/index.php?oldid=41394 *Contributors*: Jcreer

Raster Tools *Source*: http://xmswiki.com/xms/index.php?oldid=41393 *Contributors*: Jcreer

SMS Models *Source*: http://xmswiki.com/xms/index.php?oldid=39887 *Contributors*: Aanderson, Azundel, Jcreer, Rjones

FVCOM *Source*: http://xmswiki.com/xms/index.php?oldid=40153 *Contributors*: Jcreer, Jdhowlett

Generic Mesh Model *Source*: http://xmswiki.com/xms/index.php?oldid=40922 *Contributors*: Cmballard, Jcreer

Define Model Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40925 *Contributors*: Jcreer

Generic 2D Mesh Arc Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40926 *Contributors*: Jcreer, Jdhowlett

Generic 2D Nodal BC, Nodestring and Element Display Options *Source*: http://xmswiki.com/xms/index.php?oldid=40927 *Contributors*: Jcreer, Jdhowlett

Generic Model Files *Source*: http://xmswiki.com/xms/index.php?oldid=40928 *Contributors*: Jcreer, Jdhowlett

Generic Model Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=41289 *Contributors*: Jcreer, Sramiah

PTM *Source*: http://xmswiki.com/xms/index.php?oldid=40646 *Contributors*: Jcreer, Morelandtom

PTM Overview *Source*: http://xmswiki.com/xms/index.php?oldid=40286 *Contributors*: Jcreer

PTM Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40647 *Contributors*: Jcreer, Jdhowlett

PTM Particle Filters *Source*: http://xmswiki.com/xms/index.php?oldid=41287 *Contributors*: Jcreer

PTM Polygon Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40651 *Contributors*: Jcreer, Jdhowlett

PTM Traps *Source*: http://xmswiki.com/xms/index.php?oldid=40653 *Contributors*: Jcreer, Jdhowlett

PTM Coverages *Source*: http://xmswiki.com/xms/index.php?oldid=40655 *Contributors*: Jcreer

PTM Gage Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=42001 *Contributors*: Jcreer

PTM Sources *Source*: http://xmswiki.com/xms/index.php?oldid=40654 *Contributors*: Jcreer, Jdhowlett

PTM Arc Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40658 *Contributors*: Jcreer, Jdhowlett

PTM Feature Point Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40659 *Contributors*: Jcreer, Jdhowlett

PTM Files *Source*: http://xmswiki.com/xms/index.php?oldid=40660 *Contributors*: Jcreer, Morelandtom

PTM Control File *Source*: http://xmswiki.com/xms/index.php?oldid=40662 *Contributors*: Jcreer

PTM Sediment File *Source*: http://xmswiki.com/xms/index.php?oldid=40421 *Contributors*: Jcreer, Jdhowlett

PTM Source File *Source*: http://xmswiki.com/xms/index.php?oldid=40663 *Contributors*: Jcreer, Jdhowlett

PTM Wave File *Source*: http://xmswiki.com/xms/index.php?oldid=40664 *Contributors*: Jcreer, Jdhowlett

PTM Trap File *Source*: http://xmswiki.com/xms/index.php?oldid=40666 *Contributors*: Jcreer, Jdhowlett

PTM Trap Output *Source*: http://xmswiki.com/xms/index.php?oldid=42583 *Contributors*: Jcreer

PTM Boundary Condition File *Source*: http://xmswiki.com/xms/index.php?oldid=40668 *Contributors*: Jcreer

PTM Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40670 *Contributors*: Jcreer, Jdhowlett

PTM Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40671 *Contributors*: Jcreer, Jdhowlett

PTM Model Check *Source*: http://xmswiki.com/xms/index.php?oldid=40672 *Contributors*: Jcreer, Jdhowlett

CSHORE *Source*: http://xmswiki.com/xms/index.php?oldid=40151 *Contributors*: Jcreer

CSTORM-MS *Source*: http://xmswiki.com/xms/index.php?oldid=40152 *Contributors*: Jcreer

ADCIRC *Source*: http://xmswiki.com/xms/index.php?oldid=40835 *Contributors*: Jcreer, Jdhowlett

ADCIRC Database *Source*: http://xmswiki.com/xms/index.php?oldid=40816 *Contributors*: Azundel, Jcreer, Jdye

ADCIRC Files *Source*: http://xmswiki.com/xms/index.php?oldid=40817 *Contributors*: Cballard, Jcreer

ADCIRC Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40818 *Contributors*: Cballard, Jcreer

ADCIRC Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40819 *Contributors*: Jcreer, Jdhowlett

ADCIRC Mesh *Source*: http://xmswiki.com/xms/index.php?oldid=40821 *Contributors*: Jcreer, Jdhowlett

ADCIRC Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=41253 *Contributors*: Jcreer, Rjones

ADCIRC Point Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=31401 *Contributors*: Jcreer, Jdhowlett

ADCIRC Spatial Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=40824 *Contributors*: Jcreer, Rjones

ADCIRC Spatial Attributes 11.1 *Source*: http://xmswiki.com/xms/index.php?oldid=40825 *Contributors*: Jcreer

ADCIRC/CMS-Wave Steering *Source*: http://xmswiki.com/xms/index.php?oldid=41198 *Contributors*: Jcreer, Jdhowlett

LTEA *Source*: http://xmswiki.com/xms/index.php?oldid=42757 *Contributors*: Jcreer, Wood

Q&A ADCIRC *Source*: http://xmswiki.com/xms/index.php?oldid=40828 *Contributors*: Jcreer, Wood

Saving ADCIRC *Source*: http://xmswiki.com/xms/index.php?oldid=42008 *Contributors*: Astrassburg, Jcreer

ADCIRC Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=41199 *Contributors*: Jcreer, Jdhowlett

ADCIRC BC Nodestrings *Source*: http://xmswiki.com/xms/index.php?oldid=40833 *Contributors*: Jcreer

ADCIRC Weirs and Island Barriers *Source*: http://xmswiki.com/xms/index.php?oldid=40832 *Contributors*: Jcreer

BOUSS-2D *Source*: http://xmswiki.com/xms/index.php?oldid=40744 *Contributors*: Jcreer, Jdhowlett

BOUSS-2D Calculators *Source*: http://xmswiki.com/xms/index.php?oldid=40714 *Contributors*: Jcreer, Jdhowlett

BOUSS-2D Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40720 *Contributors*: Jcreer, Jdhowlett

BOUSS-2D Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40723 *Contributors*: Jcreer, Jdhowlett

BOUSS-2D Parameter File *Source*: http://xmswiki.com/xms/index.php?oldid=40724 *Contributors*: Azundel, Jcreer

BOUSS-2D Probes *Source*: http://xmswiki.com/xms/index.php?oldid=40725 *Contributors*: Jcreer

BOUSS2D Right click Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40023 *Contributors*: Jcreer

BOUSS-2D Test Cases *Source*: http://xmswiki.com/xms/index.php?oldid=40727 *Contributors*: Dmyers, Jcreer

Q&A BOUSS-2D *Source*: http://xmswiki.com/xms/index.php?oldid=40728 *Contributors*: Jcreer, Jdhowlett

Saving BOUSS-2D *Source*: http://xmswiki.com/xms/index.php?oldid=42012 *Contributors*: Astrassburg, Jcreer

BOUSS2D Runup / Overtopping *Source*: http://xmswiki.com/xms/index.php?oldid=40730 *Contributors*: Jcreer

BOUSS2D Runup / Overtopping Input Files *Source*: http://xmswiki.com/xms/index.php?oldid=40732 *Contributors*: Jcreer

BOUSS2D Runup / Overtopping Viewing Data *Source*: http://xmswiki.com/xms/index.php?oldid=40731 *Contributors*: Jcreer

CGWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=40810 *Contributors*: Azundel, Jcreer

CGWAVE Overview *Source*: http://xmswiki.com/xms/index.php?oldid=39590 *Contributors*: Jcreer

CGWAVE BC Node Strings *Source*: http://xmswiki.com/xms/index.php?oldid=40790 *Contributors*: Jcreer, Jdhowlett, Wood

CGWAVE Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40793 *Contributors*: Jcreer, Jdhowlett

CGWAVE Boundary Conditions Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40794 *Contributors*: Jcreer, Jdhowlett

CGWAVE Files *Source*: http://xmswiki.com/xms/index.php?oldid=40795 *Contributors*: Azundel, Jcreer

CGWAVE Governing Equations *Source*: http://xmswiki.com/xms/index.php?oldid=40796 *Contributors*: Jcreer, Jdhowlett

CGWAVE Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40797 *Contributors*: Azundel, Jcreer

CGWAVE Incident Wave Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40798 *Contributors*: Azundel, Jcreer

CGWAVE Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40799 *Contributors*: Jcreer, Jdhowlett

CGWAVE Model Checker *Source*: http://xmswiki.com/xms/index.php?oldid=40800 *Contributors*: Jcreer, Jdhowlett

CGWAVE Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40801 *Contributors*: Jcreer, Jdhowlett

CGWAVE Numerical Solution *Source*: http://xmswiki.com/xms/index.php?oldid=40802 *Contributors*: Jcreer, Jdhowlett

CGWAVE Practical Notes *Source*: http://xmswiki.com/xms/index.php?oldid=40803 *Contributors*: Astrassburg, Jcreer

CGWAVE Test Cases *Source*: http://xmswiki.com/xms/index.php?oldid=40804 *Contributors*: Jcreer, Jdhowlett

Long Wave Input Toolbox *Source*: http://xmswiki.com/xms/index.php?oldid=40806 *Contributors*: Azundel, Jcreer

Q&A CGWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=40807 *Contributors*: Cjenkins, Jcreer

Saving CGWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=42010 *Contributors*: Astrassburg, Jcreer

CMS *Source*: http://xmswiki.com/xms/index.php?oldid=41022 *Contributors*: Jcreer, Mebrown998

CMS-Flow/CMS-Wave Steering *Source*: http://xmswiki.com/xms/index.php?oldid=40811 *Contributors*: Jcreer, Jdhowlett

CMS-Flow/Save Points *Source*: http://xmswiki.com/xms/index.php?oldid=41036 *Contributors*: Jcreer

CMS-Flow *Source*: http://xmswiki.com/xms/index.php?oldid=41623 *Contributors*: Azundel, Jcreer, Twood, Vnarteh, Wood

CMS-Flow Cell Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=40745 *Contributors*: Jcreer, Jdhowlett

CMS-Flow Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=41044 *Contributors*: Azundel, Jcreer

CMS-Flow Grid Modification Flag *Source*: http://xmswiki.com/xms/index.php?oldid=40748 *Contributors*: Jcreer

CMS-Flow Hard Bottom *Source*: http://xmswiki.com/xms/index.php?oldid=41056 *Contributors*: Jcreer, Jdhowlett

CMS-Flow Hot Start *Source*: http://xmswiki.com/xms/index.php?oldid=41043 *Contributors*: Jcreer, Jdye

CMS-Flow Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41061 *Contributors*: Jcreer, Jdhowlett

CMS-Flow Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=41062 *Contributors*: Azundel, Jcreer

CMS-Flow Observation Cells *Source*: http://xmswiki.com/xms/index.php?oldid=40753 *Contributors*: Jcreer, Jdhowlett

CMS-Flow Output Control *Source*: http://xmswiki.com/xms/index.php?oldid=40754 *Contributors*: Azundel, Jcreer

CMS-Flow Salinity *Source*: http://xmswiki.com/xms/index.php?oldid=41028 *Contributors*: Jcreer, Jdhowlett

CMS-Flow Spatial Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=40756 *Contributors*: Azundel, Jcreer

CMS-Flow Time Control *Source*: http://xmswiki.com/xms/index.php?oldid=41029 *Contributors*: Azundel, Jcreer

CMS-Flow Transport Control *Source*: http://xmswiki.com/xms/index.php?oldid=41034 *Contributors*: Jcreer, Jschultze

CMS-Flow User-Editable Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=40759 *Contributors*: Jcreer

Lund Cirp and Watanabe Formula *Source*: http://xmswiki.com/xms/index.php?oldid=41058 *Contributors*: Azundel, Jcreer

M2D *Source*: http://xmswiki.com/xms/index.php?oldid=40762 *Contributors*: Jcreer, Mebrown998

Q&A CMS-Flow *Source*: http://xmswiki.com/xms/index.php?oldid=40257 *Contributors*: Jcreer, Jdhowlett

Telescoping Grids *Source*: http://xmswiki.com/xms/index.php?oldid=40763 *Contributors*: Jcreer

CMS-Wave *Source*: http://xmswiki.com/xms/index.php?oldid=40788 *Contributors*: Azundel, Jcreer

CMS-WAVE Cell Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40770 *Contributors*: Jcreer, Sramiah

CMS-WAVE Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40775 *Contributors*: Azundel, Jcreer

CMS-WAVE Menu *Source*: http://xmswiki.com/xms/index.php?oldid=39341 *Contributors*: Jcreer, Jdhowlett

CMS-WAVE Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=39339 *Contributors*: Jcreer, Jdhowlett

CMS-Wave Model Control Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40780 *Contributors*: Jcreer

CMS-WAVE Spectral Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40781 *Contributors*: Jcreer

CMS-WAVE Files *Source*: http://xmswiki.com/xms/index.php?oldid=42086 *Contributors*: Azundel, Jcreer

CMS-WAVE Control File *Source*: http://xmswiki.com/xms/index.php?oldid=40782 *Contributors*: Azundel, Jcreer

CMS-WAVE Simulation File *Source*: http://xmswiki.com/xms/index.php?oldid=40783 *Contributors*: Azundel, Jcreer

CMS-WAVE Spectral Energy File *Source*: http://xmswiki.com/xms/index.php?oldid=40784 *Contributors*: Jcreer, Ladamson

CMS-WAVE Spectral Table File *Source*: http://xmswiki.com/xms/index.php?oldid=40785 *Contributors*: Azundel, Jcreer

CMS-Wave STD *Source*: http://xmswiki.com/xms/index.php?oldid=40786 *Contributors*: Jcreer

CMS-WAVE Structure File *Source*: http://xmswiki.com/xms/index.php?oldid=40787 *Contributors*: Azundel, Jcreer

GenCade *Source*: http://xmswiki.com/xms/index.php?oldid=40841 *Contributors*: Jcreer

GenCade Files *Source*: http://xmswiki.com/xms/index.php?oldid=40838 *Contributors*: Jcreer

GenCade Modeling Process *Source*: http://xmswiki.com/xms/index.php?oldid=40839 *Contributors*: Jcreer

Practical Notes For Using GenCade *Source*: http://xmswiki.com/xms/index.php?oldid=40840 *Contributors*: Jcreer

GenCade Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40842 *Contributors*: Jcreer

GenCade Arc Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=40843 *Contributors*: Jcreer

GenCade Events *Source*: http://xmswiki.com/xms/index.php?oldid=40844 *Contributors*: Jcreer

GenCade Menu *Source*: http://xmswiki.com/xms/index.php?oldid=41644 *Contributors*: Jcreer

GenCade Model Control Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40845 *Contributors*: Jcreer

GenCade Result Visualization *Source*: http://xmswiki.com/xms/index.php?oldid=40847 *Contributors*: Jcreer

GenCade Structures *Source*: http://xmswiki.com/xms/index.php?oldid=41653 *Contributors*: Jcreer

Wave Gages *Source*: http://xmswiki.com/xms/index.php?oldid=41640 *Contributors*: Jcreer

STWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=40704 *Contributors*: Jcreer, Wood

Grid Nesting *Source*: http://xmswiki.com/xms/index.php?oldid=40700 *Contributors*: Jcreer

Q&A STWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=40705 *Contributors*: Jcreer, Rjones

Saving STWAVE *Source*: http://xmswiki.com/xms/index.php?oldid=42009 *Contributors*: Astrassburg, Jcreer

STWAVE Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40707 *Contributors*: Jcreer

STWAVE Cell Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=40708 *Contributors*: Jcreer, Sramiah

STWAVE Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40709 *Contributors*: Jcreer, Jhworth

STWAVE Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40710 *Contributors*: Jcreer, Jdhowlett

STWAVE Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40711 *Contributors*: Jcreer, Jdhowlett

STWAVE Output Control Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40712 *Contributors*: Jcreer, Jdhowlett

WAM *Source*: http://xmswiki.com/xms/index.php?oldid=40692 *Contributors*: Jcreer

WAM – Map to Raster Utility *Source*: http://xmswiki.com/xms/index.php?oldid=40695 *Contributors*: Jcreer

WAM Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40696 *Contributors*: Jcreer

WAM Grid Options *Source*: http://xmswiki.com/xms/index.php?oldid=41095 *Contributors*: Jcreer, Tnewbold

WAM Simulation Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40698 *Contributors*: Jcreer

WAM Spectra from STWAVE Grids *Source*: http://xmswiki.com/xms/index.php?oldid=40699 *Contributors*: Jcreer

HYDRO AS-2D *Source*: http://xmswiki.com/xms/index.php?oldid=40154 *Contributors*: Jcreer, Wood

RIVERFLO-2D *Source*: http://xmswiki.com/xms/index.php?oldid=42291 *Contributors*: Aanderson, Azundel, Jcreer

SRH-2D *Source*: http://xmswiki.com/xms/index.php?oldid=42015 *Contributors*: Jcreer, Jdhowlett

Steering *Source*: http://xmswiki.com/xms/index.php?oldid=40156 *Contributors*: Jcreer, Jdhowlett

ADH *Source*: http://xmswiki.com/xms/index.php?oldid=40488 *Contributors*: Jcreer

ADH 2D Shallow Water Flow *Source*: http://xmswiki.com/xms/index.php?oldid=40890 *Contributors*: Dmyers, Jcreer

ADH Bed Layers Assignment *Source*: http://xmswiki.com/xms/index.php?oldid=40892 *Contributors*: Jcreer

ADH Boat Definition File Cards *Source*: http://xmswiki.com/xms/index.php?oldid=41269 *Contributors*: Jcreer, Kwinters

ADH Extract WSE *Source*: http://xmswiki.com/xms/index.php?oldid=40894 *Contributors*: Jcreer

ADH Hot Start *Source*: http://xmswiki.com/xms/index.php?oldid=41255 *Contributors*: Jcreer

ADH Hot Start Initial Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40897 *Contributors*: Jcreer, Kwinters

ADH Material Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40898 *Contributors*: Jcreer, Kwinters

ADH Run Model *Source*: http://xmswiki.com/xms/index.php?oldid=40899 *Contributors*: Jcreer, Jdhowlett

ADH Sediment Transport and Bed Layers *Source*: http://xmswiki.com/xms/index.php?oldid=40900 *Contributors*: Jcreer

ADH String Structures *Source*: http://xmswiki.com/xms/index.php?oldid=40901 *Contributors*: Jcreer

ADH Time Series *Source*: http://xmswiki.com/xms/index.php?oldid=42098 *Contributors*: Jcreer

ADH Time Series Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=40902 *Contributors*: Jcreer

ADH Velocity Series Editor *Source*: http://xmswiki.com/xms/index.php?oldid=40904 *Contributors*: Jcreer

ADH Vessel Coverage *Source*: http://xmswiki.com/xms/index.php?oldid=40905 *Contributors*: Jcreer

ADH Wind Stations *Source*: http://xmswiki.com/xms/index.php?oldid=40906 *Contributors*: Jcreer

ADH Boundary Condition *Source*: http://xmswiki.com/xms/index.php?oldid=42103 *Contributors*: Jcreer, Kwinters

ADH Boundary Condition File Cards *Source*: http://xmswiki.com/xms/index.php?oldid=40908 *Contributors*: Jcreer, Kwinters

ADH Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40909 *Contributors*: Jcreer, Kwinters

ADH Model Control Model Parameters *Source*: http://xmswiki.com/xms/index.php?oldid=40910 *Contributors*: Jcreer, Kwinters

ADH Model Control Iterations *Source*: http://xmswiki.com/xms/index.php?oldid=40911 *Contributors*: Jcreer

ADH Model Control Time *Source*: http://xmswiki.com/xms/index.php?oldid=40912 *Contributors*: Jcreer, Kwinters

ADH Model Control Output *Source*: http://xmswiki.com/xms/index.php?oldid=40913 *Contributors*: Jcreer, Kwinters

ADH Model Control Global Material Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40914 *Contributors*: Jcreer, Kwinters

ADH Model Control Transport Constituents *Source*: http://xmswiki.com/xms/index.php?oldid=40915 *Contributors*: Jcreer, Kwinters

ADH Model Control Consolidation *Source*: http://xmswiki.com/xms/index.php?oldid=40916 *Contributors*: Jcreer

ADH Model Control Advanced *Source*: http://xmswiki.com/xms/index.php?oldid=40917 *Contributors*: Jcreer

ADH Model Control Solver *Source*: http://xmswiki.com/xms/index.php?oldid=40918 *Contributors*: Jcreer

Sediment Library Control *Source*: http://xmswiki.com/xms/index.php?oldid=40919 *Contributors*: Jcreer

Sediment Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40920 *Contributors*: Jcreer

C2SHORE Sediment Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40921 *Contributors*: Jcreer

FESWMS *Source*: http://xmswiki.com/xms/index.php?oldid=40676 *Contributors*: Jcreer, Jdhowlett, Vnarteh

FESWMS Arc Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40675 *Contributors*: Jcreer, Jdhowlett

FESWMS BC Nodestrings *Source*: http://xmswiki.com/xms/index.php?oldid=40677 *Contributors*: Jcreer, Jdhowlett

FESWMS Errors *Source*: http://xmswiki.com/xms/index.php?oldid=40680 *Contributors*: Jcreer, Jdye

FESWMS Executable Known Issues *Source*: http://xmswiki.com/xms/index.php?oldid=40681 *Contributors*: Jcreer, Jschultze

FESWMS Files *Source*: http://xmswiki.com/xms/index.php?oldid=40682 *Contributors*: Jcreer, Jdhowlett

FESWMS Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=41841 *Contributors*: Jcreer, Jdhowlett

FESWMS Hydraulic Structures *Source*: http://xmswiki.com/xms/index.php?oldid=40684 *Contributors*: Jcreer, Ndye

FESWMS Material Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40685 *Contributors*: Jcreer, Rjones

FESWMS Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40686 *Contributors*: Jcreer, Jdhowlett

FESWMS Model Control Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40295 *Contributors*: Disleman, Jcreer

FESWMS Point Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41278 *Contributors*: Jcreer, Jdhowlett

FESWMS Polygon Attributes Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=31328 *Contributors*: Jcreer, Jdhowlett

FESWMS Sediment Control *Source*: http://xmswiki.com/xms/index.php?oldid=40689 *Contributors*: Jcreer, Twood

FESWMS Spindown *Source*: http://xmswiki.com/xms/index.php?oldid=40690 *Contributors*: Jcreer, Jdhowlett

Saving FESWMS *Source*: http://xmswiki.com/xms/index.php?oldid=42013 *Contributors*: Astrassburg, Jcreer

TABS *Source*: http://xmswiki.com/xms/index.php?oldid=40034 *Contributors*: Jcreer, Jdhowlett

TABS Attribute Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=41502 *Contributors*: Jcreer

Total Flow Nodestring *Source*: http://xmswiki.com/xms/index.php?oldid=40366 *Contributors*: Jcreer, Jdhowlett

GFGEN *Source*: http://xmswiki.com/xms/index.php?oldid=40367 *Contributors*: Jcreer, Jdhowlett

GFGEN Executable Known Issues *Source*: http://xmswiki.com/xms/index.php?oldid=40365 *Contributors*: Jcreer, Wood

RMA2 *Source*: http://xmswiki.com/xms/index.php?oldid=40889 *Contributors*: Jcreer, Jdhowlett

Nodal Transition (Marsh Porosity) Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40869 *Contributors*: Jcreer, Jdhowlett

Rainfall Values Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40871 *Contributors*: Jcreer, Jdhowlett

RMA2 1D Control Structure *Source*: http://xmswiki.com/xms/index.php?oldid=40872 *Contributors*: Jcreer

RMA2 BC *Source*: http://xmswiki.com/xms/index.php?oldid=41281 *Contributors*: Cballard, Jcreer

RMA2 Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40875 *Contributors*: Jcreer, Sramiah

RMA2 Files *Source*: http://xmswiki.com/xms/index.php?oldid=40878 *Contributors*: Jcreer, Sramiah

RMA2 Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40881 *Contributors*: Jcreer, Sramiah

RMA2 Material Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40882 *Contributors*: Jcreer, Jdhowlett

RMA2 Menu *Source*: http://xmswiki.com/xms/index.php?oldid=40884 *Contributors*: Jcreer, Jdhowlett

RMA2 Model Control Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40885 *Contributors*: Jcreer, Jdhowlett

RMA2 Spindown *Source*: http://xmswiki.com/xms/index.php?oldid=40886 *Contributors*: Jcreer, Jdhowlett

Roughness Options Dialog *Source*: http://xmswiki.com/xms/index.php?oldid=40887 *Contributors*: Jcreer, Jdhowlett

Saving RMA2 *Source*: http://xmswiki.com/xms/index.php?oldid=42007 *Contributors*: Astrassburg, Jcreer

RMA4 *Source*: http://xmswiki.com/xms/index.php?oldid=40862 *Contributors*: Jcreer, Rjones

RMA4 BC Elements *Source*: http://xmswiki.com/xms/index.php?oldid=40853 *Contributors*: Jcreer, Jdhowlett

RMA4 BC Nodestrings *Source*: http://xmswiki.com/xms/index.php?oldid=40855 *Contributors*: Jcreer

RMA4 Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=40856 *Contributors*: Jcreer, Jdhowlett

RMA4 Element Loading *Source*: http://xmswiki.com/xms/index.php?oldid=40857 *Contributors*: Jcreer, Jdye

RMA4 Files *Source*: http://xmswiki.com/xms/index.php?oldid=40859 *Contributors*: Jcreer, Jdye

RMA4 Graphical Interface *Source*: http://xmswiki.com/xms/index.php?oldid=40860 *Contributors*: Jcreer, Jdhowlett

RMA4 Material Properties *Source*: http://xmswiki.com/xms/index.php?oldid=40863 *Contributors*: Jcreer

RMA4 Menu *Source*: http://xmswiki.com/xms/index.php?oldid=42003 *Contributors*: Jcreer, Sramiah

RMA4 Model Control *Source*: http://xmswiki.com/xms/index.php?oldid=40865 *Contributors*: Jcreer

Saving RMA4 *Source*: http://xmswiki.com/xms/index.php?oldid=40866 *Contributors*: Astrassburg, Jcreer

TUFLOW *Source*: http://xmswiki.com/xms/index.php?oldid=42613 *Contributors*: Azundel, Jcreer, Rjones

TUFLOW Overview *Source*: http://xmswiki.com/xms/index.php?oldid=40285 *Contributors*: Jcreer

Batch Runs *Source*: http://xmswiki.com/xms/index.php?oldid=40538 *Contributors*: Jcreer

Command Objects *Source*: http://xmswiki.com/xms/index.php?oldid=40551 *Contributors*: Jcreer

Define TUFLOW domain *Source*: http://xmswiki.com/xms/index.php?oldid=40550 *Contributors*: Jcreer

Q&A TUFLOW *Source*: http://xmswiki.com/xms/index.php?oldid=40552 *Contributors*: Abarney, Jcreer

Reading a TUFLOW Simulation *Source*: http://xmswiki.com/xms/index.php?oldid=40554 *Contributors*: Jcreer

Saving TUFLOW *Source*: http://xmswiki.com/xms/index.php?oldid=42005 *Contributors*: Astrassburg, Jcreer

TUFLOW 2D Geometry Components *Source*: http://xmswiki.com/xms/index.php?oldid=40555 *Contributors*: Jcreer, Rjones

TUFLOW AD *Source*: http://xmswiki.com/xms/index.php?oldid=40556 *Contributors*: Jcreer

TUFLOW Boundary Conditions *Source*: http://xmswiki.com/xms/index.php?oldid=41291 *Contributors*: Jcreer, Rjones

TUFLOW Check Files *Source*: http://xmswiki.com/xms/index.php?oldid=40559 *Contributors*: Jcreer, Rjones

TUFLOW Combining 1D and 2D Domains *Source*: http://xmswiki.com/xms/index.php?oldid=40560 *Contributors*: Jcreer, Rjones

TUFLOW Grid Options *Source*: http://xmswiki.com/xms/index.php?oldid=40561 *Contributors*: Jcreer, Rjones

TUFLOW Inlet Database *Source*: http://xmswiki.com/xms/index.php?oldid=40562 *Contributors*: Jcreer, Jdhowlett

TUFLOW Interface *Source*: http://xmswiki.com/xms/index.php?oldid=42621 *Contributors*: Jcreer, Rjones

TUFLOW Irregular Culverts *Source*: http://xmswiki.com/xms/index.php?oldid=40564 *Contributors*: Jcreer

TUFLOW Linking 2D Domains *Source*: http://xmswiki.com/xms/index.php?oldid=40565 *Contributors*: Jcreer, Rjones

TUFLOW Links *Source*: http://xmswiki.com/xms/index.php?oldid=40566 *Contributors*: Jcreer

TUFLOW Manholes *Source*: http://xmswiki.com/xms/index.php?oldid=40567 *Contributors*: Jcreer

TUFLOW Material properties *Source*: http://xmswiki.com/xms/index.php?oldid=40568 *Contributors*: Jcreer

TUFLOW Menu *Source*: http://xmswiki.com/xms/index.php?oldid=42618 *Contributors*: Azundel, Jcreer

TUFLOW Model Parameters *Source*: http://xmswiki.com/xms/index.php?oldid=40570 *Contributors*: Jcreer, Jhworth

TUFLOW Network Node SX Additions *Source*: http://xmswiki.com/xms/index.php?oldid=40569 *Contributors*: Jcreer, Jdhowlett

TUFLOW Numeric Engine *Source*: http://xmswiki.com/xms/index.php?oldid=40575 *Contributors*: Jcreer

TUFLOW Running *Source*: http://xmswiki.com/xms/index.php?oldid=40572 *Contributors*: Jcreer, Rjones

TUFLOW Simulation *Source*: http://xmswiki.com/xms/index.php?oldid=40573 *Contributors*: Jcreer

TUFLOW Viewing Results *Source*: http://xmswiki.com/xms/index.php?oldid=40574 *Contributors*: Jcreer, Rjones

Quick Tour *Source*: http://xmswiki.com/xms/index.php?oldid=39843 *Contributors*: Jcreer

Quick Tour Animation *Source*: http://xmswiki.com/xms/index.php?oldid=19014 *Contributors*: Jdhowlett

Quick Tour Background Images *Source*: http://xmswiki.com/xms/index.php?oldid=30193 *Contributors*: Jdhowlett

Quick Tour Calibration *Source*: http://xmswiki.com/xms/index.php?oldid=19008 *Contributors*: Jdhowlett

Quick Tour Calibration Tartgets *Source*: http://xmswiki.com/xms/index.php?oldid=19009 *Contributors*: Jdhowlett

Quick Tour Conceptual Models *Source*: http://xmswiki.com/xms/index.php?oldid=18991 *Contributors*: Jdhowlett

Quick Tour Coordinate Conversions *Source*: http://xmswiki.com/xms/index.php?oldid=19005 *Contributors*: Jdhowlett

Quick Tour Data Calculator *Source*: http://xmswiki.com/xms/index.php?oldid=30050 *Contributors*: Jdhowlett

Quick Tour Datasets *Source*: http://xmswiki.com/xms/index.php?oldid=19012 *Contributors*: Jdhowlett

Quick Tour Edit Strip *Source*: http://xmswiki.com/xms/index.php?oldid=18985 *Contributors*: Jdhowlett

Quick Tour FESWMS... to Learn More *Source*: http://xmswiki.com/xms/index.php?oldid=19000 *Contributors*: Jdhowlett

Quick Tour FHWA *Source*: http://xmswiki.com/xms/index.php?oldid=18999 *Contributors*: Jdhowlett

Quick Tour Feature Objects *Source*: http://xmswiki.com/xms/index.php?oldid=18992 *Contributors*: Jdhowlett

Quick Tour Feature Stamping *Source*: http://xmswiki.com/xms/index.php?oldid=18995 *Contributors*: Jdhowlett

Quick Tour Filtering *Source*: http://xmswiki.com/xms/index.php?oldid=19007 *Contributors*: Jdhowlett

Quick Tour Generic Model *Source*: http://xmswiki.com/xms/index.php?oldid=19001 *Contributors*: Jdhowlett

Quick Tour Help Strip *Source*: http://xmswiki.com/xms/index.php?oldid=18989 *Contributors*: Jdhowlett

Quick Tour Layout *Source*: http://xmswiki.com/xms/index.php?oldid=18982 *Contributors*: Jdhowlett

Quick Tour Macros *Source*: http://xmswiki.com/xms/index.php?oldid=18988 *Contributors*: Jdhowlett

Quick Tour Menus *Source*: http://xmswiki.com/xms/index.php?oldid=18983 *Contributors*: Jdhowlett

Quick Tour Model Conversion *Source*: http://xmswiki.com/xms/index.php?oldid=18996 *Contributors*: Jdhowlett

Quick Tour Models... to Learn More *Source*: http://xmswiki.com/xms/index.php?oldid=18998 *Contributors*: Jdhowlett

Quick Tour Modules *Source*: http://xmswiki.com/xms/index.php?oldid=19253 *Contributors*: Jdhowlett

Quick Tour Network Creation *Source*: http://xmswiki.com/xms/index.php?oldid=18993 *Contributors*: Jdhowlett

Quick Tour Observation Coverage... to Learn More *Source*: http://xmswiki.com/xms/index.php?oldid=19011 *Contributors*: Jdhowlett

Quick Tour Polygon Attributes *Source*: http://xmswiki.com/xms/index.php?oldid=18994 *Contributors*: Jdhowlett

Quick Tour Profiles *Source*: http://xmswiki.com/xms/index.php?oldid=19010 *Contributors*: Jdhowlett

Quick Tour Project Explorer *Source*: http://xmswiki.com/xms/index.php?oldid=18984 *Contributors*: Jdhowlett

Quick Tour Scatter Data *Source*: http://xmswiki.com/xms/index.php?oldid=19006 *Contributors*: Jdhowlett

Quick Tour Support *Source*: http://xmswiki.com/xms/index.php?oldid=19015 *Contributors*: Jdhowlett

Quick Tour Tools *Source*: http://xmswiki.com/xms/index.php?oldid=18987 *Contributors*: Jdhowlett

Quick Tour USACE-WES Models *Source*: http://xmswiki.com/xms/index.php?oldid=18997 *Contributors*: Jdhowlett

Quick Tour Visualization *Source*: http://xmswiki.com/xms/index.php?oldid=19013 *Contributors*: Jdhowlett

XMDF *Source*: http://xmswiki.com/xms/index.php?oldid=41292 *Contributors*: Admin, Jcreer

File Formats *Source*: http://xmswiki.com/xms/index.php?oldid=39912 *Contributors*: Jcreer, Jdhowlett

2D Mesh Files *.2dm *Source*: http://xmswiki.com/xms/index.php?oldid=41862 *Contributors*: Azundel, Jcreer

2D Scatter Point Files *Source*: http://xmswiki.com/xms/index.php?oldid=41885 *Contributors*: Jcreer

ARC/INFO® ASCII Grid Files *.arc *Source*: http://xmswiki.com/xms/index.php?oldid=41864 *Contributors*: Jcreer, Jdhowlett

ASCII Dataset Files *.dat *Source*: http://xmswiki.com/xms/index.php?oldid=41751 *Contributors*: Jcreer, Jdhowlett

Binary Dataset Files *.dat *Source*: http://xmswiki.com/xms/index.php?oldid=41756 *Contributors*: Aanderson, Jcreer

Boundary ID Files *Source*: http://xmswiki.com/xms/index.php?oldid=39916 *Contributors*: Jcreer

Boundary XY Files *Source*: http://xmswiki.com/xms/index.php?oldid=39917 *Contributors*: Jcreer

Coastline Files *.cst *Source*: http://xmswiki.com/xms/index.php?oldid=41866 *Contributors*: Jcreer, Jdhowlett

Color Palette Files *.pal *Source*: http://xmswiki.com/xms/index.php?oldid=41868 *Contributors*: Jcreer, Jdhowlett

Drogue Files *.pth *Source*: http://xmswiki.com/xms/index.php?oldid=41870 *Contributors*: Jcreer, Jdhowlett

File Extensions *Source*: http://xmswiki.com/xms/index.php?oldid=41879 *Contributors*: Azundel, Jcreer, Jdhowlett

Fleet Wind Files *Source*: http://xmswiki.com/xms/index.php?oldid=39922 *Contributors*: Jcreer

Importing Non-Native SMS Files *Source*: http://xmswiki.com/xms/index.php?oldid=41883 *Contributors*: Jcreer, Jdhowlett

KMZ Files *Source*: http://xmswiki.com/xms/index.php?oldid=39924 *Contributors*: Jcreer, Ksmith

LandXML Files *Source*: http://xmswiki.com/xms/index.php?oldid=39926 *Contributors*: Jcreer

Map Files *Source*: http://xmswiki.com/xms/index.php?oldid=41889 *Contributors*: Jcreer, Jdhowlett

Material Files *.mat *Source*: http://xmswiki.com/xms/index.php?oldid=41872 *Contributors*: Jcreer, Jdhowlett

Native SMS Files *Source*: http://xmswiki.com/xms/index.php?oldid=41891 *Contributors*: Jcreer, Jdhowlett

Quad4 Files *Source*: http://xmswiki.com/xms/index.php?oldid=39930 *Contributors*: Jcreer, Jdhowlett

Settings Files *.ini *Source*: http://xmswiki.com/xms/index.php?oldid=41874 *Contributors*: Jcreer, Jdhowlett

Shapefiles *Source*: http://xmswiki.com/xms/index.php?oldid=41161 *Contributors*: Jcreer, Jdhowlett

SMS Super Files *.sup *Source*: http://xmswiki.com/xms/index.php?oldid=41876 *Contributors*: Jcreer, Jdhowlett

Tabular Data Files - SHOALS *.pts *Source*: http://xmswiki.com/xms/index.php?oldid=41894 *Contributors*: Azundel, Jcreer

TIN Files *Source*: http://xmswiki.com/xms/index.php?oldid=14605 *Contributors*: Jcreer, Tdye

XY Series Files *Source*: http://xmswiki.com/xms/index.php?oldid=39933 *Contributors*: Jcreer

XY Series Editor *Source*: http://xmswiki.com/xms/index.php?oldid=30324 *Contributors*: Jdhowlett

XYZ Files *Source*: http://xmswiki.com/xms/index.php?oldid=39934 *Contributors*: Jcreer

Generic Vector/Raster Files *Source*: http://xmswiki.com/xms/index.php?oldid=39935 *Contributors*: Jcreer

Import from Web *Source*: http://xmswiki.com/xms/index.php?oldid=43434 *Contributors*: Cmsmemoe, Jcreer, Ksmith, Unjedai, Wood

Registering an Image *Source*: http://xmswiki.com/xms/index.php?oldid=39779 *Contributors*: Jcreer, Ksmith

File Import Wizard *Source*: http://xmswiki.com/xms/index.php?oldid=40066 *Contributors*: Jcreer, Jdhowlett

File Import Filter Options *Source*: http://xmswiki.com/xms/index.php?oldid=41881 *Contributors*: Jcreer, Jdhowlett

File Import Wizard Supported File Formats *Source*: http://xmswiki.com/xms/index.php?oldid=41880 *Contributors*: Jcreer, Jdhowlett

GSDA:Digital Elevation *Source*: http://xmswiki.com/xms/index.php?oldid=27666 *Contributors*: Eshaw

GSDA:Decompressing Files *Source*: http://xmswiki.com/xms/index.php?oldid=38514 *Contributors*: Eshaw, Jcreer

GSDA:Importing Bathymetry Data into SMS *Source*: http://xmswiki.com/xms/index.php?oldid=24445 *Contributors*: Eshaw

GSDA:Navigating the USGS Seamless Website *Source*: http://xmswiki.com/xms/index.php?oldid=24525 *Contributors*: Eshaw

GSDA:Obtaining a DEM from USGS *Source*: http://xmswiki.com/xms/index.php?oldid=37877 *Contributors*: Cmsmemoe, Eshaw

GSDA:Obtaining Bathymetry Data from Geodas Design-a-Grid *Source*: http://xmswiki.com/xms/index.php?oldid=24593 *Contributors*: Eshaw

GSDA:Obtaining Estuarine Bathytmetry Data from NOAA's NOS *Source*: http://xmswiki.com/xms/index.php?oldid=24640 *Contributors*: Cmsmemoe

GSDA:Using GIS Data Depot to Obtain DEM, DRG and DOQ Data *Source*: http://xmswiki.com/xms/index.php?oldid=38543 *Contributors*: Jcreer

GSDA:Using the USGS Geographic Names Information System (GNIS) *Source*: http://xmswiki.com/xms/index.php?oldid=38519 *Contributors*: Cmsmemoe, Eshaw, Jcreer

GSDA:Using the USGS Map Locator to Find 7.5' Quadrangles *Source*: http://xmswiki.com/xms/index.php?oldid=24580 *Contributors*: Eshaw

GSDA:Hydrography Data *Source*: http://xmswiki.com/xms/index.php?oldid=27708 *Contributors*: Eshaw

GSDA:Obtaining Hydrographic Data from ESRI Census TIGER 1995 *Source*: http://xmswiki.com/xms/index.php?oldid=25135 *Contributors*: Eshaw

GSDA:Obtaining Hydrographic Data from the USGS / Epa *Source*: http://xmswiki.com/xms/index.php?oldid=38546 *Contributors*: Jcreer

GSDA:Obtaining Hydrographic Data from WebGIS *Source*: http://xmswiki.com/xms/index.php?oldid=38547 *Contributors*: Eshaw, Jcreer

GSDA:Obtaining Stream Data from the USGS WaterWatch *Source*: http://xmswiki.com/xms/index.php?oldid=25026 *Contributors*: Eshaw

GSDA:Obtaining Stream Stage Data from the USDA/NRCS *Source*: http://xmswiki.com/xms/index.php?oldid=38551 *Contributors*: Jcreer

GSDA:Obtaining Stream Stage Data from the USGS NWISWeb *Source*: http://xmswiki.com/xms/index.php?oldid=38548 *Contributors*: Eshaw, Jcreer

GSDA:Imagery *Source*: http://xmswiki.com/xms/index.php?oldid=27736 *Contributors*: Eshaw

GSDA:Obtaining DOQ Image Data from MSN TerraServer *Source*: http://xmswiki.com/xms/index.php?oldid=38552 *Contributors*: Eshaw, Jcreer

GSDA:Obtaining DOQs from The National Map on SDDS *Source*: http://xmswiki.com/xms/index.php?oldid=38553 *Contributors*: Eshaw, Jcreer

GSDA:Obtaining Image Data from MSN TerraServer *Source*: http://xmswiki.com/xms/index.php?oldid=24627 *Contributors*: Eshaw

GSDA:Meteorologic Data *Source*: http://xmswiki.com/xms/index.php?oldid=38554 *Contributors*: Eshaw, Jcreer

GSDA:Obtainig NEXRAD Radar Data from NCDC *Source*: http://xmswiki.com/xms/index.php?oldid=38564 *Contributors*: Jcreer

GSDA:Obtaining Precipitation Data from NOAA *Source*: http://xmswiki.com/xms/index.php?oldid=24980 *Contributors*: Eshaw

GSDA:Obtaining Precipitation Data from PRISM *Source*: http://xmswiki.com/xms/index.php?oldid=24999 *Contributors*: Eshaw

GSDA:Obtaining Precipitation Data from the NCDC / NNDC Climate Data Online *Source*: http://xmswiki.com/xms/index.php?oldid=24976 *Contributors*: Eshaw

GSDA:Obtaining Precipitation Data from the RCC *Source*: http://xmswiki.com/xms/index.php?oldid=38560 *Contributors*: Jcreer

GSDA:Obtaining Precipitation Data from the UCC *Source*: http://xmswiki.com/xms/index.php?oldid=24997 *Contributors*: Eshaw

GSDA:Obtaining Precipitation or Snow Data from USDA / NRCS *Source*: http://xmswiki.com/xms/index.php?oldid=25002 *Contributors*: Eshaw

GSDA:Oceanic Data *Source*: http://xmswiki.com/xms/index.php?oldid=27748 *Contributors*: Eshaw

GSDA:Obtaining Coastline Data from NOAA NGDC *Source*: http://xmswiki.com/xms/index.php?oldid=24598 *Contributors*: Eshaw

GSDA:Surface Characteristics *Source*: http://xmswiki.com/xms/index.php?oldid=27769 *Contributors*: Eshaw

GSDA:Creating a Table of Runoff Coefficients *Source*: http://xmswiki.com/xms/index.php?oldid=24946 *Contributors*: Eshaw

GSDA:Land Use Codes *Source*: http://xmswiki.com/xms/index.php?oldid=24800 *Contributors*: Eshaw

GSDA:Obtaining Land Use Data from the EPA *Source*: http://xmswiki.com/xms/index.php?oldid=24812 *Contributors*: Eshaw

GSDA:Obtaining Land Use Data from the USGS *Source*: http://xmswiki.com/xms/index.php?oldid=24880 *Contributors*: Eshaw

GSDA:Obtaining Land Use Data from WebGIS *Source*: http://xmswiki.com/xms/index.php?oldid=38568 *Contributors*: Eshaw, Jcreer

GSDA:Obtaining Soil Type (SSURGO) Data from Soil Data Mart *Source*: http://xmswiki.com/xms/index.php?oldid=24927 *Contributors*: Eshaw

GSDA:Obtaining Soil Type (SSURGO) Data from the Geospatial Data Gateway *Source*: http://xmswiki.com/xms/index.php?oldid=36010 *Contributors*: Eshaw, Wood

GSDA:Obtaining Soil Type (STATSGO) Data from the NRCS *Source*: http://xmswiki.com/xms/index.php?oldid=24926 *Contributors*: Eshaw

GSDA:Obtaining Soil Type Data from the NRCS - SSURGO *Source*: http://xmswiki.com/xms/index.php?oldid=38572 *Contributors*: Jcreer, Mpaudel

GSDA:Obtaining Soil Type Data from the NRCS - STATSGO *Source*: http://xmswiki.com/xms/index.php?oldid=38573 *Contributors*: Eshaw, Jcreer

GSDA:Table of Runoff Curve Numbers *Source*: http://xmswiki.com/xms/index.php?oldid=24804 *Contributors*: Eshaw

GSDA:Using Soil Type Data with GSSHA *Source*: http://xmswiki.com/xms/index.php?oldid=27822 *Contributors*: Rkent

GSDA:Obtaining Soil Type Data from the EPA *Source*: http://xmswiki.com/xms/index.php?oldid=24878 *Contributors*: Eshaw

GSDA:Statsgolmport readme *Source*: http://xmswiki.com/xms/index.php?oldid=38575 *Contributors*: Jcreer

Archived Models *Source*: http://xmswiki.com/xms/index.php?oldid=42081 *Contributors*: Jcreer

What's New in SMS 11.0 *Source*: http://xmswiki.com/xms/index.php?oldid=42850 *Contributors*: Azundel, Jcreer

What's New in SMS 10.1 *Source*: http://xmswiki.com/xms/index.php?oldid=42218 *Contributors*: Jcreer, Ladamson

What's New in SMS 10.0 *Source*: http://xmswiki.com/xms/index.php?oldid=39808 *Contributors*: Jcreer, Jdhowlett

Image Sources, Licenses and Contributors

Image:SMS Layout.png *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Layout.png *License*: unknown *Contributors*: Rjones
Image:Module toolbar.png *Source*: http://xmswiki.com/xms/index.php?title=File:Module_toolbar.png *License*: unknown *Contributors*: Rjones
Image:mesh_module.png *Source*: http://xmswiki.com/xms/index.php?title=File:Mesh_module.png *License*: unknown *Contributors*: Rjones
Image:Cartesian Grid.png *Source*: http://xmswiki.com/xms/index.php?title=File:Cartesian_Grid.png *License*: unknown *Contributors*: Rjones
Image:Scatter Set Example.png *Source*: http://xmswiki.com/xms/index.php?title=File:Scatter_Set_Example.png *License*: unknown *Contributors*: Rjones
Image:GIS Data in SMS.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GIS_Data_in_SMS.jpg *License*: unknown *Contributors*: Azundel
Image:Map_Data_Set.png *Source*: http://xmswiki.com/xms/index.php?title=File:Map_Data_Set.png *License*: unknown *Contributors*: Rjones
file:Ptm particle set.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Ptm_particle_set.jpg *License*: unknown *Contributors*: Jcreer
Image:Generic Model Interface.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Generic_Model_Interface.jpg *License*: unknown *Contributors*: Azundel
Image:SMS color filled contours.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_color_filled_contours.jpg *License*: unknown *Contributors*: Rjones
file:Vectorvisual.png *Source*: http://xmswiki.com/xms/index.php?title=File:Vectorvisual.png *License*: unknown *Contributors*: Jcreer
Image:ProfilePlot.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ProfilePlot.gif *License*: unknown *Contributors*: Abarney
file:Functionalsurface.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Functionalsurface.bmp *License*: unknown *Contributors*: Jcreer
file:Flow_trace.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Flow_trace.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS_Data_Calculator.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Data_Calculator.jpg *License*: unknown *Contributors*: Jdhowlett
Image:CreateDS.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CreateDS.jpg *License*: unknown *Contributors*: Jhworth
Image:Transform.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Transform.jpg *License*: unknown *Contributors*: Rjones
Image:CoordTrans.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CoordTrans.jpg *License*: unknown *Contributors*: Morelandtom
Image:ImportWizard2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ImportWizard2.jpg *License*: unknown *Contributors*: Morelandtom
Image:SMS image transparency.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_image_transparency.jpg *License*: unknown *Contributors*: Morelandtom
Image:CAD_Data.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CAD_Data.jpg *License*: unknown *Contributors*: Morelandtom
Image:GoogleEarth.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GoogleEarth.jpg *License*: unknown *Contributors*: Morelandtom
Image:2D_Mesh_Poly_Properties.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:2D_Mesh_Poly_Properties.jpg *License*: unknown *Contributors*: Morelandtom
Image:Terraserver.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Terraserver.jpg *License*: unknown *Contributors*: Morelandtom
Image:ZonalClass.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ZonalClass.jpg *License*: unknown *Contributors*: Morelandtom
File:SMSscreen.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMSscreen.jpg *License*: unknown *Contributors*: Jcreer
Image:FHWA_ReflectionBackground.png *Source*: http://xmswiki.com/xms/index.php?title=File:FHWA_ReflectionBackground.png *License*: unknown *Contributors*: Cbarlow
Image:Measure_tool.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Measure_tool.bmp *License*: unknown *Contributors*: Sramiah
Image:Get_data_tool.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Get_data_tool.bmp *License*: unknown *Contributors*: Sramiah
Image:Statusbar1.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Statusbar1.gif *License*: unknown *Contributors*: Abarney
Image:Statusbar2.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Statusbar2.gif *License*: unknown *Contributors*: Abarney
Image:Optional Macros Toolbar.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Optional_Macros_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Shading Options.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Shading_Options.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Contour Options.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Contour_Options.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Vector Options.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Vector_Options.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Get Module Info.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Get_Module_Info.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Plot Wizard.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Plot_Wizard.bmp *License*: unknown *Contributors*: Jdhowlett
Image:File Toolbar.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:File_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Open.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Open.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Save Project.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Save_Project.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Print.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Print.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Delete.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Delete.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Display Toolbar.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Display_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Refresh.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Refresh.jpg *License*: unknown *Contributors*: Abarney
Image:Frame.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Frame.JPG *License*: unknown *Contributors*: Jdhowlett
Image:dispopts.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:PlanView.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:PlanView.jpg *License*: unknown *Contributors*: Abarney
Image:SMS_2D_Mesh_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_2D_Mesh_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_Cartesian_Grid_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Cartesian_Grid_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_3D_Cartesian_Grid_Module_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_3D_Cartesian_Grid_Module_Icon.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_Scatter_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Scatter_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_Map_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Map_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_GIS_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_GIS_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_1D-River_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_1D-River_Module.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_1D_Grid.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_1D_Grid.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS_Particle_Module.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Particle_Module.bmp *License*: unknown *Contributors*: Jdhowlett
File:Open_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Open_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Save_Project_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Save_Project_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Get_Module_Info_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Get_Module_Info_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Print_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Print_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Delete_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Delete_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Refresh_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Refresh_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Frame_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Frame_icon.bmp *License*: unknown *Contributors*: Jcreer
File:PlanView_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:PlanView_icon.bmp *License*: unknown *Contributors*: Jcreer
File:FrontView_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:FrontView_icon.bmp *License*: unknown *Contributors*: Jcreer
File:ObliqueView_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:ObliqueView_icon.bmp *License*: unknown *Contributors*: Jcreer
File:SideView_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SideView_icon.bmp *License*: unknown *Contributors*: Jcreer
File:PreviousView_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:PreviousView_icon.bmp *License*: unknown *Contributors*: Jcreer

File:PlotWizard_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:PlotWizard_icon.bmp *License*: unknown *Contributors*: Jcreer
Image:Static Tools Toolbar.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Static_Tools_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Tool_Pan.GIF *Source*: http://xmswiki.com/xms/index.php?title=File:Tool_Pan.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Zoom.GIF *Source*: http://xmswiki.com/xms/index.php?title=File:Tool_Zoom.GIF *License*: unknown *Contributors*: Jhandy
Image:Tool_Rotate.GIF *Source*: http://xmswiki.com/xms/index.php?title=File:Tool_Rotate.GIF *License*: unknown *Contributors*: Jhandy
File:Time Step Window.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Time_Step_Window.jpg *License*: unknown *Contributors*: Jcreer
Image:SMS Module Toolbar.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Module_Toolbar.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Breakline.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Breakline.gif *License*: unknown *Contributors*: Cballard
Image:ComputedVsObserved.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ComputedVsObserved.gif *License*: unknown *Contributors*: Abarney
Image:ErrorSummary.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ErrorSummary.gif *License*: unknown *Contributors*: Abarney
Image:ErrorVsSimulation.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ErrorVsSimulation.gif *License*: unknown *Contributors*: Abarney
Image:ErrorVsTimestep.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ErrorVsTimestep.gif *License*: unknown *Contributors*: Abarney
Image:Plot3.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Plot3.JPG *License*: unknown *Contributors*: Astrassburg
Image:Plot4.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Plot4.JPG *License*: unknown *Contributors*: Astrassburg
Image:ResidualVsObserved.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ResidualVsObserved.gif *License*: unknown *Contributors*: Abarney
Image:TimeSeriesEditor.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:TimeSeriesEditor.jpg *License*: unknown *Contributors*: Jdhowlett
Image:TimeSeries.gif *Source*: http://xmswiki.com/xms/index.php?title=File:TimeSeries.gif *License*: unknown *Contributors*: Abarney
File:elem_angles.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_angles.png *License*: unknown *Contributors*: Jcreer
File:BaryDomain.png *Source*: http://xmswiki.com/xms/index.php?title=File:BaryDomain.png *License*: unknown *Contributors*: Jcreer
File:elem_angles_ordered.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_angles_ordered.png *License*: unknown *Contributors*: Jcreer
File:Contours.png *Source*: http://xmswiki.com/xms/index.php?title=File:Contours.png *License*: unknown *Contributors*: Jcreer
Image:Bama.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Bama.gif *License*: unknown *Contributors*: Cballard
Image:Noyo.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Noyo.gif *License*: unknown *Contributors*: Cballard
Image:ellipsiodcoordsys.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Ellipsiodcoordsys.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Cppcoordsys1.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Cppcoordsys2.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys3.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Cppcoordsys3.jpg *License*: unknown *Contributors*: Ksmith
Image:cppcoordsys4.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Cppcoordsys4.jpg *License*: unknown *Contributors*: Ksmith
Image:Africa.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Africa.gif *License*: unknown *Contributors*: Cballard
Image:asia.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Asia.gif *License*: unknown *Contributors*: Cballard
Image:Australia.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Australia.gif *License*: unknown *Contributors*: Cballard
Image:Usutm.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Usutm.gif *License*: unknown *Contributors*: Jdhowlett
Image:UTM_world_no_Image_Map.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:UTM_world_no_Image_Map.jpg *License*: unknown *Contributors*: Abarney
Image:Europe.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Europe.gif *License*: unknown *Contributors*: Cballard
Image:Northamerica.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Northamerica.gif *License*: unknown *Contributors*: Cballard
Image:Southamerica.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Southamerica.gif *License*: unknown *Contributors*: Cballard
Image:USstate_plane_no_image_map.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:USstate_plane_no_image_map.jpg *License*: unknown *Contributors*: Abarney
Image:Ak27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Ak27.gif *License*: unknown *Contributors*: Cballard
Image:Ha27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Ha27.gif *License*: unknown *Contributors*: Cballard
Image:Me27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Me27.gif *License*: unknown *Contributors*: Cballard
Image:Mw27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Mw27.gif *License*: unknown *Contributors*: Cballard
Image:Ne27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Ne27.gif *License*: unknown *Contributors*: Cballard
Image:Nw27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Nw27.gif *License*: unknown *Contributors*: Cballard
Image:Sc27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Sc27.gif *License*: unknown *Contributors*: Cballard
Image:Se27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Se27.gif *License*: unknown *Contributors*: Cballard
Image:Sw27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Sw27.gif *License*: unknown *Contributors*: Cballard
Image:Virg27.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Virg27.gif *License*: unknown *Contributors*: Cballard
Image:SMS Inactive Elevation Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Elevation_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS Active Elevation Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Elevation_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS Inactive Scalar Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Scalar_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS Active Scalar Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Scalar_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS Inactive Vector Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Vector_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:SMS Active Vector Data Set.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Vector_Data_Set.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Smooth_Data_Sets.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Smooth_Data_Sets.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Dispopts.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Dispopts.gif *License*: unknown *Contributors*: Jdhowlett
Image:LightingDialog.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:LightingDialog.jpg *License*: unknown *Contributors*: Ksmith
Image:RasterDialog.png *Source*: http://xmswiki.com/xms/index.php?title=File:RasterDialog.png *License*: unknown *Contributors*: Cdickson
Image:step33f.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Step33f.gif *License*: unknown *Contributors*: Trabadi
Image:step44.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Step44.gif *License*: unknown *Contributors*: Trabadi
Image:step55.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Step55.gif *License*: unknown *Contributors*: Trabadi
Image:step66.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Step66.gif *License*: unknown *Contributors*: Trabadi
Image:step77.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Step77.gif *License*: unknown *Contributors*: Trabadi
Image:ImportFromWeb-CurrentCoordSystemWarning.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-CurrentCoordSystemWarning.png *License*: unknown *Contributors*: Unjedai
Image:ImportFromWeb-VirtualEarthMapLocator.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-VirtualEarthMapLocator.png *License*: unknown *Contributors*: Unjedai
Image:ImportFromWeb-WebServiceOptions.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-WebServiceOptions.png *License*: unknown *Contributors*: Cmsmemoe, Ksmith, Unjedai
Image:ImportFromWeb-Save.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-Save.png *License*: unknown *Contributors*: Unjedai
Image:ImportFromWeb-CreateFiles.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-CreateFiles.png *License*: unknown *Contributors*: Unjedai
Image:ImportFromWeb-Initializing.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-Initializing.png *License*: unknown *Contributors*: Unjedai
Image:ImportFromWeb-SelectScale.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-SelectScale.png *License*: unknown *Contributors*: Unjedai

Image:ImportFromWeb-TilesRemaining.png *Source*: http://xmswiki.com/xms/index.php?title=File:ImportFromWeb-TilesRemaining.png *License*: unknown *Contributors*: Unjedai

Image:SelectRegistrationPoint.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SelectRegistrationPoint.jpg *License*: unknown *Contributors*: Abarney

Image:Zoom.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Zoom.JPG *License*: unknown *Contributors*: Jdhowlett

Image:Pan.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Pan.JPG *License*: unknown *Contributors*: Jdhowlett

Image:Save_as_image.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Save_as_image.bmp *License*: unknown *Contributors*: Sramiah

Image:SelectCGridCell.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:SelectCGridCell.JPG *License*: unknown *Contributors*: Abarney

Image:Rotate.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Rotate.JPG *License*: unknown *Contributors*: Jdhowlett

Image:contours.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Contours.gif *License*: unknown *Contributors*: Abarney

File:SMS_Inactive_Elevation_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Elevation_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SMS_Active_Elevation_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Elevation_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SMS_Inactive_Scalar_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Scalar_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SMS_Active_Scalar_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Scalar_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SMS_Inactive_Vector_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Inactive_Vector_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SMS_Active_Vector_Data_Set_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_Active_Vector_Data_Set_icon.bmp *License*: unknown *Contributors*: Jcreer

Image:AnnotationTree.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AnnotationTree.jpg *License*: unknown *Contributors*: Djordan

Image:Scale.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Scale.jpg *License*: unknown *Contributors*: Sramiah

Image:North-arrow.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:North-arrow.jpg *License*: unknown *Contributors*: Ksmith

Image:Select Point.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Point.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Create Point.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Point.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Select Detached Breakwater.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Detached_Breakwater.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Create Breakwater.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Breakwater.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Select Jetty or Groin.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Jetty_or_Groin.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Create Jetty Groin.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Jetty_Groin.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Select Seawall.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Seawall.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Create Seawall.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Seawall.bmp *License*: unknown *Contributors*: Jdhowlett

Image:1DGridDataTree.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:1DGridDataTree.JPG *License*: unknown *Contributors*: Abarney

Image:Riv_profile_plot.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Riv_profile_plot.gif *License*: unknown *Contributors*: Abarney

Image:Select Cross-section.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Cross-section.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Select Centerline.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Centerline.bmp *License*: unknown *Contributors*: Jdhowlett

Image:cross_section_editor.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Cross_section_editor.jpg *License*: unknown *Contributors*: Abarney

Image:GridType1.gif *Source*: http://xmswiki.com/xms/index.php?title=File:GridType1.gif *License*: unknown *Contributors*: Jhandy

File:SMS_CGrid_Select_Cell_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Cell_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Select_Row_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Row_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Select_Column_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Column_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Split_Grid_Column_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Split_Grid_Column_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Split_Grid_Row_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Split_Grid_Row_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Drag_Column_Boundary_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Drag_Column_Boundary_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Drag_Row_Boundary_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Drag_Row_Boundary_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:SMS_CGrid_Select_Cell_String_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Cell_String_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:CGridDataTree.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:CGridDataTree.JPG *License*: unknown *Contributors*: Abarney

Image:SMS_CGrid_Select_Cell_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Cell_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Select_Row_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Row_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Select_Column_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Column_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Split_Grid_Column_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Split_Grid_Column_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Split_Grid_Row_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Split_Grid_Row_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Drag_Column_Boundary_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Drag_Column_Boundary_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Drag_Row_Boundary_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Drag_Row_Boundary_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

Image:SMS_CGrid_Select_Cell_String_Tool_Icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS_CGrid_Select_Cell_String_Tool_Icon.bmp *License*: unknown *Contributors*: Jdhowlett

File:CreateMeshNodestring_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:CreateMeshNodestring_icon.bmp *License*: unknown *Contributors*: Jcreer

File:SelectMeshNodestring.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SelectMeshNodestring.bmp *License*: unknown *Contributors*: Jcreer

Image:Select ArcObject.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_ArcObject.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Get ArcObject Attributes.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Get_ArcObject_Attributes.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Select Object.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Object.bmp *License*: unknown *Contributors*: Jdhowlett

Image:Add Data Dialog.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Add_Data_Dialog.jpg *License*: unknown *Contributors*: Jdhowlett

Image:Join Table.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Join_Table.jpg *License*: unknown *Contributors*: Jdhowlett

Image:Map Properties.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Map_Properties.jpg *License*: unknown *Contributors*: Jdhowlett

Image:GIS to Feature Objects Wizard -- Step 1 of 2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GIS_to_Feature_Objects_Wizard_--_Step_1_of_2.jpg *License*: unknown *Contributors*: Jdhowlett

Image:Map Data Set.png *Source*: http://xmswiki.com/xms/index.php?title=File:Map_Data_Set.png *License*: unknown *Contributors*: Rjones

File:Select_Feature_Vertex_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Vertex_icon.bmp *License*: unknown *Contributors*: Jcreer

File:Create_Feature_Vertex_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Feature_Vertex_icon.bmp *License*: unknown *Contributors*: Jcreer

File:Select_Feature_Arc_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Arc_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Create_Feature_Arc_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Feature_Arc_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Select_Feature_Arc_Group_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Arc_Group_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Select_Feature_Polygon_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Polygon_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Select_1D_Grid_Frame_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_1D_Grid_Frame_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Create_2D_Grid_Frame_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_2D_Grid_Frame_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Select_2D_Grid_Frame_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_2D_Grid_Frame_icon.bmp *License*: unknown *Contributors*: Jcreer
Image:Project_explorer_coverages.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Project_explorer_coverages.jpg *License*: unknown *Contributors*: Abarney
Image:StampedGuidebankPoints.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:StampedGuidebankPoints.jpg *License*: unknown *Contributors*: Azundel
Image:StampedCenterLine.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:StampedCenterLine.jpg *License*: unknown *Contributors*: Azundel
Image:Guidebank.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Guidebank.jpg *License*: unknown *Contributors*: Azundel
Image:WingWalls.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:WingWalls.jpg *License*: unknown *Contributors*: Azundel
Image:Spillthrough.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Spillthrough.jpg *License*: unknown *Contributors*: Azundel
Image:StampedGuidebankArcs plan.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:StampedGuidebankArcs_plan.jpg *License*: unknown *Contributors*: Azundel
Image:StampedGuidebankArcs.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:StampedGuidebankArcs.jpg *License*: unknown *Contributors*: Azundel
Image:Observ_arcs1.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Observ_arcs1.JPG *License*: unknown *Contributors*: Astrassburg
Image:Calibration_targets.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Calibration_targets.JPG *License*: unknown *Contributors*: Astrassburg
Image:SelectCompassPlotTool.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SelectCompassPlotTool.jpg *License*: unknown *Contributors*: Jdhowlett
Image:CompassPlotPropertiesDlg.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CompassPlotPropertiesDlg.jpg *License*: unknown *Contributors*: Jdhowlett
File:1Dhydcrosssec.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:1Dhydcrosssec.bmp *License*: unknown *Contributors*: Jcreer
Image:River_Conceptual Model.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:River_Conceptual_Model.jpg *License*: unknown *Contributors*: Abarney
File:Stormpath1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath1.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath2.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath3.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath3.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath4.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath4.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath5.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath5.jpg *License*: unknown *Contributors*: Jcreer
File:Stormpath6.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Stormpath6.jpg *License*: unknown *Contributors*: Jcreer
File:SyntheticPBLCentralPressureGraph.png *Source*: http://xmswiki.com/xms/index.php?title=File:SyntheticPBLCentralPressureGraph.png *License*: unknown *Contributors*: Jcreer
File:FC_poly_attributes.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:FC_poly_attributes.jpg *License*: unknown *Contributors*: Jdhowlett
File:FC_node_attributes_and_bridge.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:FC_node_attributes_and_bridge.jpg *License*: unknown *Contributors*: Jdhowlett
File:Bridge_chart.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Bridge_chart.jpg *License*: unknown *Contributors*: Jdhowlett
File:400px-ZShape_failing_levee.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:400px-ZShape_failing_levee.bmp *License*: unknown *Contributors*: Jcreer
File:zshape_property_dlg.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Zshape_property_dlg.jpg *License*: unknown *Contributors*: Jcreer
File:200px-ZShape_trigger_point_dialog.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:200px-ZShape_trigger_point_dialog.bmp *License*: unknown *Contributors*: Jcreer
Image:MapDataTree.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:MapDataTree.jpg *License*: unknown *Contributors*: Abarney
Image:Select Feature Point or Node.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Point_or_Node.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Create Feature Point.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Feature_Point.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select Feature Vertex.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Vertex.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Create Feature Vertex.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Feature_Vertex.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select Feature Arc.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Arc.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Create Feature Arc.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Feature_Arc.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select Feature Arc Group.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Arc_Group.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select Feature Polygon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_Feature_Polygon.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select 1D Grid Frame.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_1D_Grid_Frame.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Create 2D Grid Frame.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_2D_Grid_Frame.bmp *License*: unknown *Contributors*: Jdhowlett
Image:Select 2D Grid Frame.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Select_2D_Grid_Frame.bmp *License*: unknown *Contributors*: Jdhowlett
Image:featureobjects.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Featureobjects.gif *License*: unknown *Contributors*: Cballard
Image:Polygons.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Polygons.gif *License*: unknown *Contributors*: Cballard
File:Clean.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Clean.jpg *License*: unknown *Contributors*: Jcreer
File:Redistvert.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Redistvert.jpg *License*: unknown *Contributors*: Jcreer
File:BeforeSourceArcRedistribution.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:BeforeSourceArcRedistribution.jpg *License*: unknown *Contributors*: Jcreer
File:AfterSourceArcRedistribution.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AfterSourceArcRedistribution.jpg *License*: unknown *Contributors*: Jcreer
Image:TypesOfMeshElements.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:TypesOfMeshElements.jpg *License*: unknown *Contributors*: Jdhowlett
Image:TriangulateExample.gif *Source*: http://xmswiki.com/xms/index.php?title=File:TriangulateExample.gif *License*: unknown *Contributors*: Abarney
Image:TriangleCircumcircle.gif *Source*: http://xmswiki.com/xms/index.php?title=File:TriangleCircumcircle.gif *License*: unknown *Contributors*: Abarney
Image:Patch_Mesh_Polygon.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Patch_Mesh_Polygon.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Rectangular_patch.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Rectangular_patch.gif *License*: unknown *Contributors*: Abarney
Image:Triangular_patch.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Triangular_patch.gif *License*: unknown *Contributors*: Abarney
Image:create_node.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Create_node.gif *License*: unknown *Contributors*: Jdhowlett
Image:select_node.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Select_node.gif *License*: unknown *Contributors*: Jdhowlett
Image:CreateMeshNodestring.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:CreateMeshNodestring.JPG *License*: unknown *Contributors*: Jdhowlett
Image:SelectMeshNodestring.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:SelectMeshNodestring.JPG *License*: unknown *Contributors*: Jdhowlett
Image:line2.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Line2.gif *License*: unknown *Contributors*: Jdhowlett
Image:tri3.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Tri3.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad4.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Quad4.gif *License*: unknown *Contributors*: Jdhowlett
Image:line3.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Line3.gif *License*: unknown *Contributors*: Jdhowlett
Image:tri6.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Tri6.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad8.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Quad8.gif *License*: unknown *Contributors*: Jdhowlett
Image:quad9.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Quad9.gif *License*: unknown *Contributors*: Jdhowlett
Image:SelectMeshElement.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SelectMeshElement.gif *License*: unknown *Contributors*: Jdhowlett
Image:swap_edge.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Swap_edge.gif *License*: unknown *Contributors*: Jdhowlett

Image:split_merge.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Split_merge.gif *License*: unknown *Contributors*: Jdhowlett
Image:contour_label.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Contour_label.gif *License*: unknown *Contributors*: Jdhowlett
Image:elem_1d_linear.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_1d_linear.png *License*: unknown *Contributors*: Unjedai
Image:elem_1d_quad.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_1d_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_transition.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_transition.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_tri.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_2d_linear_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_tri.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_2d_quad_tri.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_linear_quad.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_2d_linear_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad.png *License*: unknown *Contributors*: Unjedai
Image:elem_2d_quad_quad_wcenter.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_2d_quad_quad_wcenter.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_tet.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_3d_linear_tet.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_prism.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_3d_linear_prism.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_hex.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_3d_linear_hex.png *License*: unknown *Contributors*: Unjedai
Image:elem_3d_linear_pyramid.png *Source*: http://xmswiki.com/xms/index.php?title=File:Elem_3d_linear_pyramid.png *License*: unknown *Contributors*: Unjedai
Image:Breaklineexample.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Breaklineexample.gif *License*: unknown *Contributors*: Abarney
Image:Smoothnodestringexample.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Smoothnodestringexample.gif *License*: unknown *Contributors*: Abarney
Image:RefineTypesExample.gif *Source*: http://xmswiki.com/xms/index.php?title=File:RefineTypesExample.gif *License*: unknown *Contributors*: Abarney
Image:Refineelements.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Refineelements.gif *License*: unknown *Contributors*: Abarney
File:InterpNodesExample.gif *Source*: http://xmswiki.com/xms/index.php?title=File:InterpNodesExample.gif *License*: unknown *Contributors*: Abarney
File:NewNodePattern.png *Source*: http://xmswiki.com/xms/index.php?title=File:NewNodePattern.png *License*: unknown *Contributors*: Azundel
File:SMS-Particle_grid_data_set_bin_elevations_101_datum.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS-Particle_grid_data_set_bin_elevations_101_datum.bmp *License*: unknown *Contributors*: Jcreer
File:SMS-Particle_grid_data_set_bin_elevations_101_datum_z.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS-Particle_grid_data_set_bin_elevations_101_datum_z.bmp *License*: unknown *Contributors*: Jcreer
File:SMS-Particle_grid_data_set_bin_elevations_101_datum_bathy.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS-Particle_grid_data_set_bin_elevations_101_datum_bathy.bmp *License*: unknown *Contributors*: Jcreer
File:SMS-Particle_grid_data_set_bin_elevations_101_datum_wse.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:SMS-Particle_grid_data_set_bin_elevations_101_datum_wse.bmp *License*: unknown *Contributors*: Jcreer
Image:SMS-Particle_Module_Create_Data_Sets.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMS-Particle_Module_Create_Data_Sets.jpg *License*: unknown *Contributors*: Jdhowlett
Image:SelectScatterPoint.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SelectScatterPoint.jpg *License*: unknown *Contributors*: Cballard
Image:SelectMeshElement.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SelectMeshElement.jpg *License*: unknown *Contributors*: Cballard
Image:CreateLinearMeshTriangle.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CreateLinearMeshTriangle.jpg *License*: unknown *Contributors*: Cballard
Image:SwapEdgeMesh.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SwapEdgeMesh.jpg *License*: unknown *Contributors*: Cballard
Image:Triangulation_optimization.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Triangulation_optimization.jpg *License*: unknown *Contributors*: Abarney
File:Triangulate1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Triangulate1.jpg *License*: unknown *Contributors*: Abarney
File:Triangulate2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Triangulate2.jpg *License*: unknown *Contributors*: Abarney
File:DelaunayCriterion.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:DelaunayCriterion.jpg *License*: unknown *Contributors*: Jcreer
Image:Check_long_triangles.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Check_long_triangles.jpg *License*: unknown *Contributors*: Abarney
Image:SelectScatterPoint.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:SelectScatterPoint.JPG *License*: unknown *Contributors*: Jdhowlett
Image:gridframe.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Gridframe.gif *License*: unknown *Contributors*: Abarney
Image:Scatter Breakline Options.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Scatter_Breakline_Options.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Process_Boundary_Triangles_Dialog.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Process_Boundary_Triangles_Dialog.jpg *License*: unknown *Contributors*: Jdhowlett
Image:Scatter_Set_Bow_Tie_Triangles.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Scatter_Set_Bow_Tie_Triangles.jpg *License*: unknown *Contributors*: Jdhowlett
File:Autobreaklines.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Autobreaklines.jpg *License*: unknown *Contributors*: Jcreer
File:Filter_diagram.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Filter_diagram.bmp *License*: unknown *Contributors*: Jcreer
File:LaplacianInterpolation zoomed in.png *Source*: http://xmswiki.com/xms/index.php?title=File:LaplacianInterpolation_zoomed_in.png *License*: unknown *Contributors*: Jcreer
File:LaplacianInterp.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:LaplacianInterp.jpg *License*: unknown *Contributors*: Jcreer
Image:Equation30.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Equation30.JPG *License*: unknown *Contributors*: Abarney
Image:Equation31.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Equation31.JPG *License*: unknown *Contributors*: Abarney
Image:Equation32.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Equation32.JPG *License*: unknown *Contributors*: Abarney
Image:Equation33.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Equation33.JPG *License*: unknown *Contributors*: Abarney
Image:Four_quadrants.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Four_quadrants.jpg *License*: unknown *Contributors*: Abarney
Image:Enclosing_triangle.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Enclosing_triangle.jpg *License*: unknown *Contributors*: Abarney
Image:Equation34.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation34.jpg *License*: unknown *Contributors*: Abarney
Image:Barycentric_coords.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Barycentric_coords.jpg *License*: unknown *Contributors*: Abarney
Image:Equation35.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation35.jpg *License*: unknown *Contributors*: Abarney
Image:Equation36.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation36.jpg *License*: unknown *Contributors*: Abarney
Image:Equation37.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation37.jpg *License*: unknown *Contributors*: Abarney
Image:Equation38.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation38.jpg *License*: unknown *Contributors*: Abarney
Image:Equation39.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation39.jpg *License*: unknown *Contributors*: Abarney
Image:Delauney_triangulation.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Delauney_triangulation.jpg *License*: unknown *Contributors*: Abarney
Image:Equation41.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation41.jpg *License*: unknown *Contributors*: Abarney
Image:Thiessen_polygons.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Thiessen_polygons.jpg *License*: unknown *Contributors*: Abarney
File:ProfilePlotRaster.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:ProfilePlotRaster.bmp *License*: unknown *Contributors*: Jcreer
File:ConverSelectedRasterToScatter.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:ConverSelectedRasterToScatter.bmp *License*: unknown *Contributors*: Jcreer
File:define model dialog.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Define_model_dialog.jpg *License*: unknown *Contributors*: Jcreer
Image:LeProvost_Domain.png *Source*: http://xmswiki.com/xms/index.php?title=File:LeProvost_Domain.png *License*: unknown *Contributors*: Mebrown998
Image:Spatial atts.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Spatial_atts.JPG *License*: unknown *Contributors*: Astrassburg
Image:Spatial atts2.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:Spatial_atts2.JPG *License*: unknown *Contributors*: Astrassburg
File:SpatialAttsDialog.PNG *Source*: http://xmswiki.com/xms/index.php?title=File:SpatialAttsDialog.PNG *License*: unknown *Contributors*: Jcreer
File:SpatialAttsInitialValues.PNG *Source*: http://xmswiki.com/xms/index.php?title=File:SpatialAttsInitialValues.PNG *License*: unknown *Contributors*: Jcreer

Image:AmericanSamoaLT12K.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaLT12K.jpg *License*: unknown *Contributors*: Azundel
Image:AmericanSamoaPaved.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaPaved.jpg *License*: unknown *Contributors*: Azundel
Image:AmericanSamoaLT24K.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaLT24K.jpg *License*: unknown *Contributors*: Azundel
Image:AmericanSamoaLT12K_Zoom.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaLT12K_Zoom.jpg *License*: unknown *Contributors*: Azundel
Image:AmericanSamoaPaved_Zoom.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaPaved_Zoom.jpg *License*: unknown *Contributors*: Azundel
Image:AmericanSamoaLT24K_Zoom.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:AmericanSamoaLT24K_Zoom.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBayImage.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBayImage.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBayConceptual.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBayConceptual.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBayLiinear.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBayLiinear.jpg *License*: unknown *Contributors*: Azundel
Image:LTEA_Harmonics.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:LTEA_Harmonics.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBay30K.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBay30K.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBay10K.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBay10K.jpg *License*: unknown *Contributors*: Azundel
Image:GlacierBay20K.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:GlacierBay20K.jpg *License*: unknown *Contributors*: Azundel
File:Effective_length2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Effective_length2.jpg *License*: unknown *Contributors*: Wood
File:adcirc nodestrings.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Adcirc_nodestrings.jpg *License*: unknown *Contributors*: Jcreer
File:ADCIRCweir1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ADCIRCweir1.jpg *License*: unknown *Contributors*: Jcreer
File:ADCIRCweir2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ADCIRCweir2.jpg *License*: unknown *Contributors*: Jcreer
File:ADCIRCweir4.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ADCIRCweir4.jpg *License*: unknown *Contributors*: Jcreer
File:ADCIRCWeir5.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ADCIRCWeir5.jpg *License*: unknown *Contributors*: Jcreer
File:ADCIRCweir6.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:ADCIRCweir6.jpg *License*: unknown *Contributors*: Jcreer
Image:Damping Reflection Relationship-Fig 5- BOUSS-2D Technical Report.png *Source*: http://xmswiki.com/xms/index.php?title=File:Damping_Reflection_Relationship-Fig_5-_BOUSS-2D_Technical_Report.png *License*: unknown *Contributors*: Azundel
File:probe manager.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Probe_manager.jpg *License*: unknown *Contributors*: Jcreer
File:TransectProfile.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:TransectProfile.jpg *License*: unknown *Contributors*: Jcreer
File:SolutionProfile.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SolutionProfile.jpg *License*: unknown *Contributors*: Jcreer
File:SolutionTimeSeries.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SolutionTimeSeries.jpg *License*: unknown *Contributors*: Jcreer
Image:CGWave_figure1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CGWave_figure1.jpg *License*: unknown *Contributors*: Abarney
Image:Dialog1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Dialog1.jpg *License*: unknown *Contributors*: Abarney
Image:Dialog2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Dialog2.jpg *License*: unknown *Contributors*: Abarney
Image:Equation12.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation12.jpg *License*: unknown *Contributors*: Abarney
Image:alpha.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Alpha.jpg *License*: unknown *Contributors*: Abarney
Image:Equation13.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation13.jpg *License*: unknown *Contributors*: Abarney
Image:Equation14.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation14.jpg *License*: unknown *Contributors*: Abarney
Image:Theta.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Theta.jpg *License*: unknown *Contributors*: Abarney
Image:Beta.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Beta.jpg *License*: unknown *Contributors*: Abarney
Image:Equation15.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation15.jpg *License*: unknown *Contributors*: Abarney
Image:Equation16.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation16.jpg *License*: unknown *Contributors*: Abarney
Image:Chi.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Chi.jpg *License*: unknown *Contributors*: Abarney
Image:Equation17.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation17.jpg *License*: unknown *Contributors*: Abarney
Image:Equation18.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation18.jpg *License*: unknown *Contributors*: Abarney
Image:Equation19.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation19.jpg *License*: unknown *Contributors*: Abarney
Image:Equation20.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation20.jpg *License*: unknown *Contributors*: Abarney
Image:Equation21.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation21.jpg *License*: unknown *Contributors*: Abarney
Image:Equation22.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation22.jpg *License*: unknown *Contributors*: Abarney
Image:Equation23.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation23.jpg *License*: unknown *Contributors*: Abarney
Image:Equation24.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation24.jpg *License*: unknown *Contributors*: Abarney
Image:Equation25.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation25.jpg *License*: unknown *Contributors*: Abarney
Image:Equation26.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation26.jpg *License*: unknown *Contributors*: Abarney
Image:Equation27.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation27.jpg *License*: unknown *Contributors*: Abarney
Image:Equation1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation1.jpg *License*: unknown *Contributors*: Abarney
Image:Equation2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation2.jpg *License*: unknown *Contributors*: Abarney
Image:Equation3.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation3.jpg *License*: unknown *Contributors*: Abarney
Image:Equation4.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation4.jpg *License*: unknown *Contributors*: Abarney
Image:Sigma.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Sigma.jpg *License*: unknown *Contributors*: Abarney
Image:Equation5.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation5.jpg *License*: unknown *Contributors*: Abarney
Image:pi_.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Pi_.jpg *License*: unknown *Contributors*: Abarney
Image:sigma.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Sigma.jpg *License*: unknown *Contributors*: Abarney
Image:phi.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Phi.jpg *License*: unknown *Contributors*: Abarney
Image:Equation7.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation7.jpg *License*: unknown *Contributors*: Abarney
Image:Equation8.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation8.jpg *License*: unknown *Contributors*: Abarney
Image:Equation9.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation9.jpg *License*: unknown *Contributors*: Abarney
Image:Equation10.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation10.jpg *License*: unknown *Contributors*: Abarney
Image:Equation11.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Equation11.jpg *License*: unknown *Contributors*: Abarney
File:561px-CMSFlowModelControl.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:561px-CMSFlowModelControl.jpg *License*: unknown *Contributors*: Jcreer
File:CustomDatasets.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:CustomDatasets.jpg *License*: unknown *Contributors*: Jcreer
File:Create_Breakwater_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Breakwater_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Create_Seawall_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Seawall_icon.bmp *License*: unknown *Contributors*: Jcreer
File:Create_Jetty_Groin_icon.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Create_Jetty_Groin_icon.bmp *License*: unknown *Contributors*: Jcreer
Image:Right arrow.png *Source*: http://xmswiki.com/xms/index.php?title=File:Right_arrow.png *License*: unknown *Contributors*: Kwinters
File:300px-ADH_Bed_Layer_Spec_Conflict.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:300px-ADH_Bed_Layer_Spec_Conflict.bmp *License*: unknown *Contributors*: Jcreer
Image:culvert.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Culvert.jpg *License*: unknown *Contributors*: Cjenkins
Image:wier.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Wier.jpg *License*: unknown *Contributors*: Cjenkins

Image:TotalFlow1.gif *Source*: http://xmswiki.com/xms/index.php?title=File:TotalFlow1.gif *License*: unknown *Contributors*: Jdhowlett
Image:TotalFlow2.gif *Source*: http://xmswiki.com/xms/index.php?title=File:TotalFlow2.gif *License*: unknown *Contributors*: Jdhowlett
File:200px-RMA2_Transition_Element.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:200px-RMA2_Transition_Element.bmp *License*: unknown *Contributors*: Jcreer
File:100px-Culvert.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:100px-Culvert.bmp *License*: unknown *Contributors*: Jcreer
File:Wier.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:Wier.bmp *License*: unknown *Contributors*: Jcreer
Image:DisFactors.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:DisFactors.JPG *License*: unknown *Contributors*: Cballard
Image:RMA2_BC.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:RMA2_BC.jpg *License*: unknown *Contributors*: Cballard
Image:NodeBC.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:NodeBC.JPG *License*: unknown *Contributors*: Cballard
File:rma4 bc nodestrings.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Rma4_bc_nodestrings.jpg *License*: unknown *Contributors*: Jcreer
File:RMA4_Assign_Element_Mass_Loading.JPG *Source*: http://xmswiki.com/xms/index.php?title=File:RMA4_Assign_Element_Mass_Loading.JPG *License*: unknown *Contributors*: Jdye
File:rma4 material properties.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Rma4_material_properties.jpg *License*: unknown *Contributors*: Jcreer
File:rma4 model control dialog 1.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Rma4_model_control_dialog_1.jpg *License*: unknown *Contributors*: Jcreer
Image:TUFLOW 1D 2D linking mechanisms.png *Source*: http://xmswiki.com/xms/index.php?title=File:TUFLOW_1D_2D_linking_mechanisms.png *License*: unknown *Contributors*: Rjones
Image:TUFLOW transitioning1Dcross 2Ddomain.png *Source*: http://xmswiki.com/xms/index.php?title=File:TUFLOW_transitioning1Dcross_2Ddomain.png *License*: unknown *Contributors*: Rjones
Image:2Dto2DLinkage.png *Source*: http://xmswiki.com/xms/index.php?title=File:2Dto2DLinkage.png *License*: unknown *Contributors*: Rjones
File:tuflow links.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Tuflow_links.jpg *License*: unknown *Contributors*: Jcreer
Image:Quick_Tour.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Quick_Tour.jpg *License*: unknown *Contributors*: Abarney
Image:noyo.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Noyo.gif *License*: unknown *Contributors*: Cballard
Image:bama.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Bama.gif *License*: unknown *Contributors*: Cballard
Image:image3.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image3.jpg *License*: unknown *Contributors*: Cballard
Image:image4.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image4.jpg *License*: unknown *Contributors*: Cballard
Image:compobsplot.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Compobsplot.gif *License*: unknown *Contributors*: Cballard
Image:image20.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image20.jpg *License*: unknown *Contributors*: Cballard
Image:coord.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Coord.gif *License*: unknown *Contributors*: Cballard
Image:datacalc.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Datacalc.gif *License*: unknown *Contributors*: Cballard
Image:databrowser.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Databrowser.gif *License*: unknown *Contributors*: Cballard
Image:image10.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Image10.gif *License*: unknown *Contributors*: Cballard
Image:image13.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image13.jpg *License*: unknown *Contributors*: Cballard
Image:featstamp.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Featstamp.gif *License*: unknown *Contributors*: Cballard
Image:image6a.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Image6a.gif *License*: unknown *Contributors*: Cballard
Image:image5.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image5.jpg *License*: unknown *Contributors*: Cballard
Image:image6.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image6.jpg *License*: unknown *Contributors*: Cballard
Image:image7.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image7.jpg *License*: unknown *Contributors*: Cballard
Image:image8.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image8.jpg *License*: unknown *Contributors*: Cballard
Image:image9.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image9.jpg *License*: unknown *Contributors*: Cballard
Image:image14.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Image14.gif *License*: unknown *Contributors*: Cballard
Image:image2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image2.jpg *License*: unknown *Contributors*: Cballard
Image:macros.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Macros.gif *License*: unknown *Contributors*: Cballard
Image:SMSMenus.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMSMenus.jpg *License*: unknown *Contributors*: Abarney
Image:image17.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image17.jpg *License*: unknown *Contributors*: Cballard
Image:SMSModules.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:SMSModules.jpg *License*: unknown *Contributors*: Abarney
Image:Meshmodule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Meshmodule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMSCGridModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMSCGridModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:3DCGridModule.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:3DCGridModule.jpg *License*: unknown *Contributors*: Jdhowlett
Image:ScatterModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:ScatterModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMSMapModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMSMapModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMSGISModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMSGISModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMS1DRiverModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMS1DRiverModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMS1DGridModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMS1DGridModule.gif *License*: unknown *Contributors*: Jdhowlett
Image:SMSParticleModule.gif *Source*: http://xmswiki.com/xms/index.php?title=File:SMSParticleModule.gif *License*: unknown *Contributors*: Abarney
Image:image14.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image14.jpg *License*: unknown *Contributors*: Cballard
Image:polydlog.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Polydlog.gif *License*: unknown *Contributors*: Cballard
Image:profileplot.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Profileplot.gif *License*: unknown *Contributors*: Cballard
Image:image9.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Image9.gif *License*: unknown *Contributors*: Cballard
Image:tools.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Tools.gif *License*: unknown *Contributors*: Cballard
Image:image19.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Image19.jpg *License*: unknown *Contributors*: Cballard
Image:image011.gif *Source*: http://xmswiki.com/xms/index.php?title=File:Image011.gif *License*: unknown *Contributors*: Cballard
Image:Fleet Wind Dialog.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:Fleet_Wind_Dialog.jpg *License*: unknown *Contributors*: Jcreer
Image:GoogleEarth.png *Source*: http://xmswiki.com/xms/index.php?title=File:GoogleEarth.png *License*: unknown *Contributors*: Unjedai
Image:File Import Wizard - Step 1 of 2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:File_Import_Wizard_-_Step_1_of_2.jpg *License*: unknown *Contributors*: Jdhowlett
Image:File Import Wizard - Step 2 of 2.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:File_Import_Wizard_-_Step_2_of_2.jpg *License*: unknown *Contributors*: Jdhowlett
Image:File Import Filter Options.jpg *Source*: http://xmswiki.com/xms/index.php?title=File:File_Import_Filter_Options.jpg *License*: unknown *Contributors*: Jdhowlett
File:GSDA_USGS.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_USGS.png *License*: unknown *Contributors*: Cmsmemoe
File:lpdaac.png *Source*: http://xmswiki.com/xms/index.php?title=File:Lpdaac.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_webGIS.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_webGIS.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_GeoBase.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_GeoBase.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_CGIAR.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_CGIAR.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_MapMart.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_MapMart.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_GeoCommunity.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_GeoCommunity.png *License*: unknown *Contributors*: Cmsmemoe
File:GSDA_LandInfo.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA_LandInfo.png *License*: unknown *Contributors*: Cmsmemoe

Image:GSDAViewandDownloadUnitedStatesData.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAViewandDownloadUnitedStatesData.png *License*: unknown *Contributors*: Eshaw
Image:GSDAZoomToolbar.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAZoomToolbar.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomIn.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomIn.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomFullExtent.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomFullExtent.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomOut.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomOut.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessScaleInformation.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessScaleInformation.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomRegion.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomRegion.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomPrevious.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomPrevious.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomPoint.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomPoint.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessQueryToolbar.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessQueryToolbar.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessMapAtt.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessMapAtt.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessGeographicPlace.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessGeographicPlace.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessElevation.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessElevation.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessUSGridt.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessUSGridt.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessElevProfile.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessElevProfile.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessToolsTools.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessToolsTools.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessPan.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessPan.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessMeasure.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessMeasure.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessClearMeasure.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessClearMeasure.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessCreatePDF.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessCreatePDF.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessMetadata.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessMetadata.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessLayerTransparency.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessLayerTransparency.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessDownloadTools.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessDownloadTools.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessDefineDownload.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessDefineDownload.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessDefineAreaByCoordinates.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessDefineAreaByCoordinates.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessDocumentTools.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessDocumentTools.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessMapInfoMetadata.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessMapInfoMetadata.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessLegend.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessLegend.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessMapRefresh.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessMapRefresh.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessUserInstructions.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessUserInstructions.png *License*: unknown *Contributors*: Eshaw
Image:GSDASeamlessZoomTool.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDASeamlessZoomTool.png *License*: unknown *Contributors*: Eshaw
Image:GSDAExtractBoxTool.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAExtractBoxTool.png *License*: unknown *Contributors*: Eshaw
Image:DEM1ArcSec.png *Source*: http://xmswiki.com/xms/index.php?title=File:DEM1ArcSec.png *License*: unknown *Contributors*: Eshaw
File:GSDAObtainingDEMData.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAObtainingDEMData.bmp *License*: unknown *Contributors*: Jcreer
File:GSDADoylestownOut.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDADoylestownOut.bmp *License*: unknown *Contributors*: Jcreer
File:GSDADoylestownIn.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDADoylestownIn.bmp *License*: unknown *Contributors*: Jcreer
File:GSDAObtainingDOQData.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAObtainingDOQData.bmp *License*: unknown *Contributors*: Jcreer
Image:GNISTable.png *Source*: http://xmswiki.com/xms/index.php?title=File:GNISTable.png *License*: unknown *Contributors*: Eshaw
Image:GSDAMapLocator.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAMapLocator.png *License*: unknown *Contributors*: Eshaw
Image:GSDA7.5Quad.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA7.5Quad.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage075.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage075.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage076.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage076.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage077.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage077.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage087.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage087.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage088.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage088.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage084.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage084.png *License*: unknown *Contributors*: Eshaw
File:GSDAImage085.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage085.bmp *License*: unknown *Contributors*: Jcreer
File:GSDAImage086.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage086.bmp *License*: unknown *Contributors*: Jcreer
File:GSDAImage073.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage073.bmp *License*: unknown *Contributors*: Jcreer
Image:GSDAImage072.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage072.png *License*: unknown *Contributors*: Eshaw
Image:GSDADraper7.5Quad.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDADraper7.5Quad.png *License*: unknown *Contributors*: Eshaw
Image:GSDAWharfDOQ.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAWharfDOQ.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage001.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage001.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage002.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage002.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage003.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage003.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage004.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage004.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage005.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage005.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage006.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage006.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage007.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage007.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage008.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage008.png *License*: unknown *Contributors*: Eshaw
Image:GSDATerraServerSLC.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDATerraServerSLC.png *License*: unknown *Contributors*: Eshaw
Image:GSDATerraServerDeltaCenter.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDATerraServerDeltaCenter.png *License*: unknown *Contributors*: Eshaw
Image:GSDA057.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA057.png *License*: unknown *Contributors*: Eshaw
Image:GSDA058.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA058.png *License*: unknown *Contributors*: Eshaw
Image:GSDA059.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA059.png *License*: unknown *Contributors*: Eshaw
Image:GSDA060.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA060.png *License*: unknown *Contributors*: Eshaw
Image:GSDA061.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDA061.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage054.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage054.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage055.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage055.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage056.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage056.png *License*: unknown *Contributors*: Eshaw

Image:GSDAImage066.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage066.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage067.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage067.png *License*: unknown *Contributors*: Eshaw
File:GSDAImage062.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage062.bmp *License*: unknown *Contributors*: Jcreer
File:GSDAImage063.bmp *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage063.bmp *License*: unknown *Contributors*: Jcreer
Image:GSDAImage064.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage064.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage065.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage065.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage068.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage068.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage069.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage069.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage070.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage070.png *License*: unknown *Contributors*: Eshaw
Image:NGDC.png *Source*: http://xmswiki.com/xms/index.php?title=File:NGDC.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage041.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage041.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage042.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage042.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage035.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage035.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage040.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage040.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage024.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage024.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage044.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage044.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage023.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage023.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage043.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage043.png *License*: unknown *Contributors*: Eshaw
Image:GSDAImage045.png *Source*: http://xmswiki.com/xms/index.php?title=File:GSDAImage045.png *License*: unknown *Contributors*: Eshaw